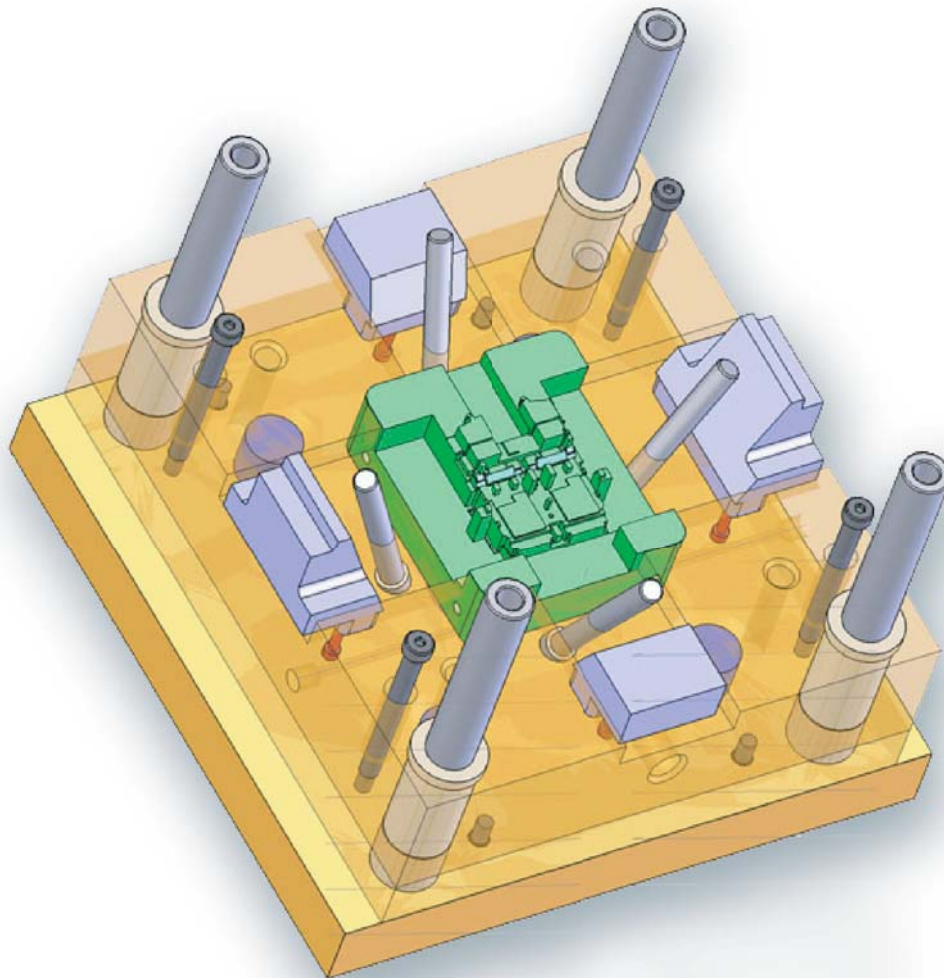


3DQuickMould – User Manual



To Enable Tooling Engineers To Use The Power of 3D Design

Contents

A. PREPRATION	7
B. SOFTWARE INTERFACE	9
C. FUNCTIONS.....	13
CHAPTER 1. DESIGN MODULES AND TOOLS	13
1.PRODUCT SHRINKAGE	13
2.START PROJECT	17
3.WORKPIECE	18
4. PRODUCT ASSEMBLY	24
5.LAYOUT MANAGER	27
6.MOLDBASE MANAGER	52
7.EJECTOR MANAGER.....	78
1.Position.....	78
2.Add ejectors	84
3.Trim/Pocket	87
4.Edit Ejectors	92
8.COOLING MANAGER	94
1. Path.....	95

2. Parameters.....	124
3. Edit	126
4. Accessory	127
9.LIBRARY MANAGER.....	130
10.UNDERCUT MANAGER	137
1.Components	138
2.Standard lifter	160
3.Standard slide.....	160
4.Libraries.....	161
11.SUBINSERT MANAGER.....	162
1. Define body	164
2. Create holder	173
3. Pocket	178
12.FEED MANAGER	179
1.References	180
2.Horizontal runner	184
3.Vertical runner.....	185
4.Cold slug well.....	192
6.Gate design	187
13.ELECTRODE MANAGER	205
1.Derive part.....	206
2.Define body	210

2.Create holder	215
4.Tools	219

CHAPTER 2. DESIGN UTILITIES ERROR! BOOKMARK NOT DEFINED.

1.SOLID UTILITIESERROR! BOOKMARK NOT DEFINED.

1. Information.....	Error! Bookmark not defined.
2. Solid patch	Error! Bookmark not defined.
3. Solid Operation	Error! Bookmark not defined.
4. Quick replace.....	Error! Bookmark not defined.
5. Solid attribute.....	Error! Bookmark not defined.

2.SURFACE UTILITIES ...ERROR! BOOKMARK NOT DEFINED.

1.Cut relationship.....	Error! Bookmark not defined.
2.Untrim surface	Error! Bookmark not defined.
3.Multiple copy	Error! Bookmark not defined.
4.Composite loft.....	Error! Bookmark not defined.
5.Quick loft	Error! Bookmark not defined.
6.Quick plane.....	Error! Bookmark not defined.
7.Prepare-trimming	Error! Bookmark not defined.

3.MOLD TOOLSERROR! BOOKMARK NOT DEFINED.

1.Pocketing.....	Error! Bookmark not defined.
------------------	------------------------------

2.Grouping.....	Error! Bookmark not defined.
3.Paste body	Error! Bookmark not defined.
4.Save project	Error! Bookmark not defined.
5.Mold motion	Error! Bookmark not defined.
6.Pocket corner.....	Error! Bookmark not defined.
7.Clearance	Error! Bookmark not defined.
8.Set configuration	Error! Bookmark not defined.

CHAPTER 3. SELECTION TOOLS .. **ERROR! BOOKMARK NOT DEFINED.**

1.CONTINUOUS EDGES ERROR! BOOKMARK NOT DEFINED.

2.BOX SELECTIONERROR! BOOKMARK NOT DEFINED.

3.FACE SEARCHERROR! BOOKMARK NOT DEFINED.

CHAPTER 4. VIEW TOOLS ...**ERROR! BOOKMARK NOT DEFINED.**

1.CUSTOM VIEW.....ERROR! BOOKMARK NOT DEFINED.

2.TOGGLE HIDE/SHOW .ERROR! BOOKMARK NOT DEFINED.

3.ACTIVATE WORKING MODEL ERROR!
BOOKMARK NOT DEFINED.

4.TWO PERPENDICULAR ITEMS?..... ERROR!
BOOKMARK NOT DEFINED.

CHAPTER 5. ENTITY GROUPING.. ERROR! BOOKMARK
NOT DEFINED.

1.CAVITY FACES.....ERROR! BOOKMARK NOT
DEFINED.

2.CORE FACES.....ERROR! BOOKMARK NOT
DEFINED.

3.OUTER EDGESERROR! BOOKMARK NOT
DEFINED.

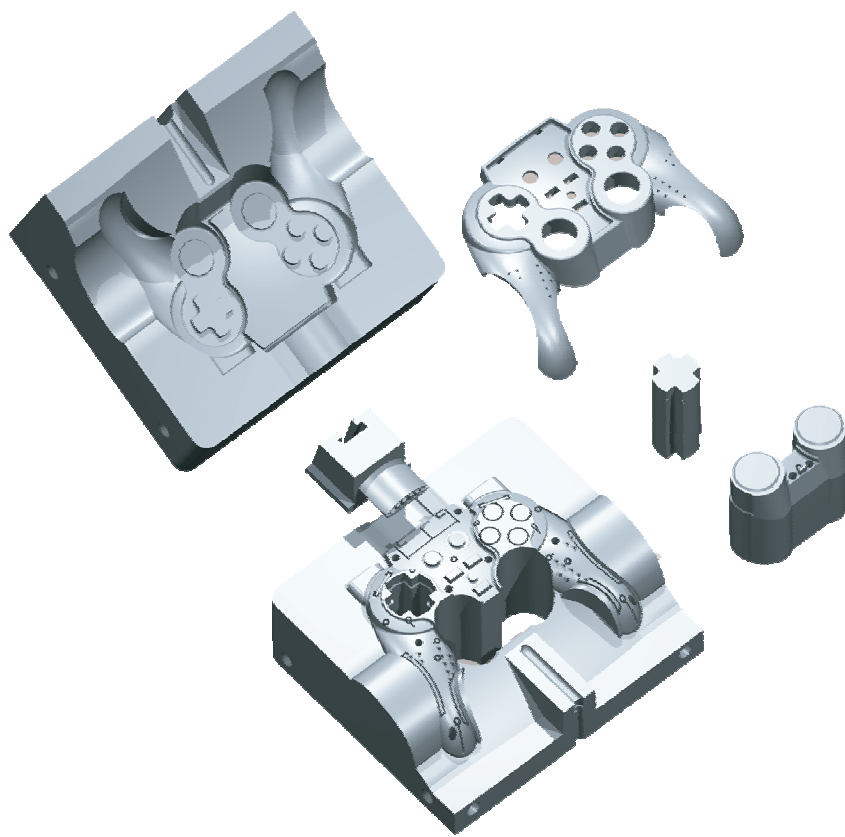
4.INNER EDGES.....ERROR! BOOKMARK NOT
DEFINED.

5.PATCHING SURFACES..... ERROR! BOOKMARK
NOT DEFINED.

6.PARTING SURFACES .ERROR! BOOKMARK NOT
DEFINED.

CHAPTER 6. OTHER TOOLS .ERROR! BOOKMARK NOT
DEFINED.

CHAPTER 7. STANDARD PARTS IN 3DQUICKMOLD
.....ERROR! BOOKMARK NOT DEFINED.

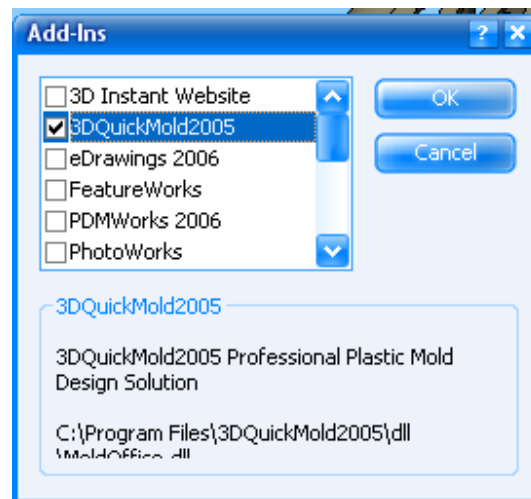


A. Preparation

3DQuickMold is professional plastic mold design software seamlessly integrated with **Solidworks**. This manual is based on the platform Solidworks 2006.

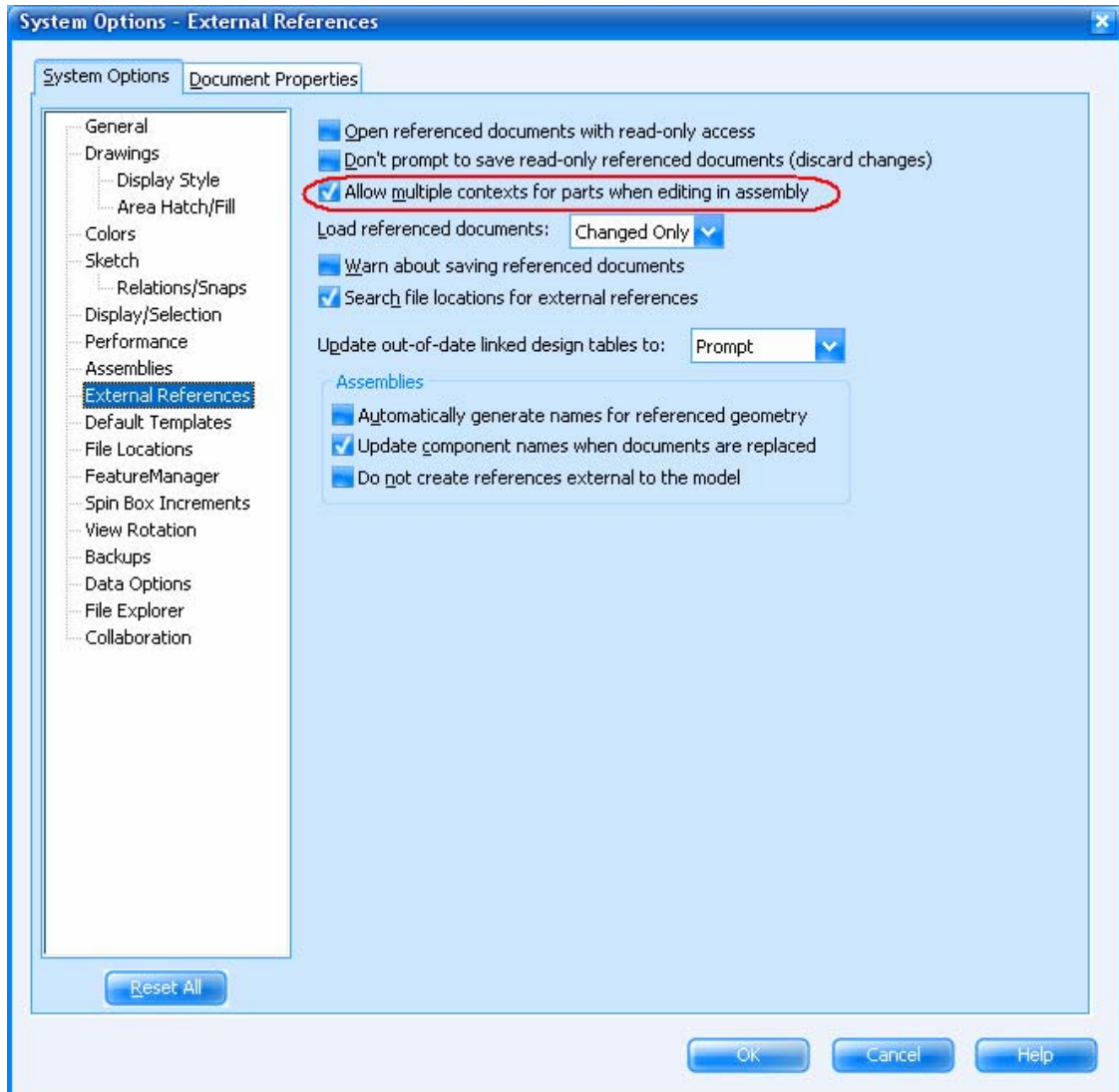
Please do the following after the installation:

1. Open Tools-> Add-Ins, Select 3DQuickMold,



2. Select the Option as follows:
Tools -> Options->System Options ->External References ->Allow multiple contexts for parts when editing in assembly

For the latest edition, this option is selected when 3DQuickMold is started.





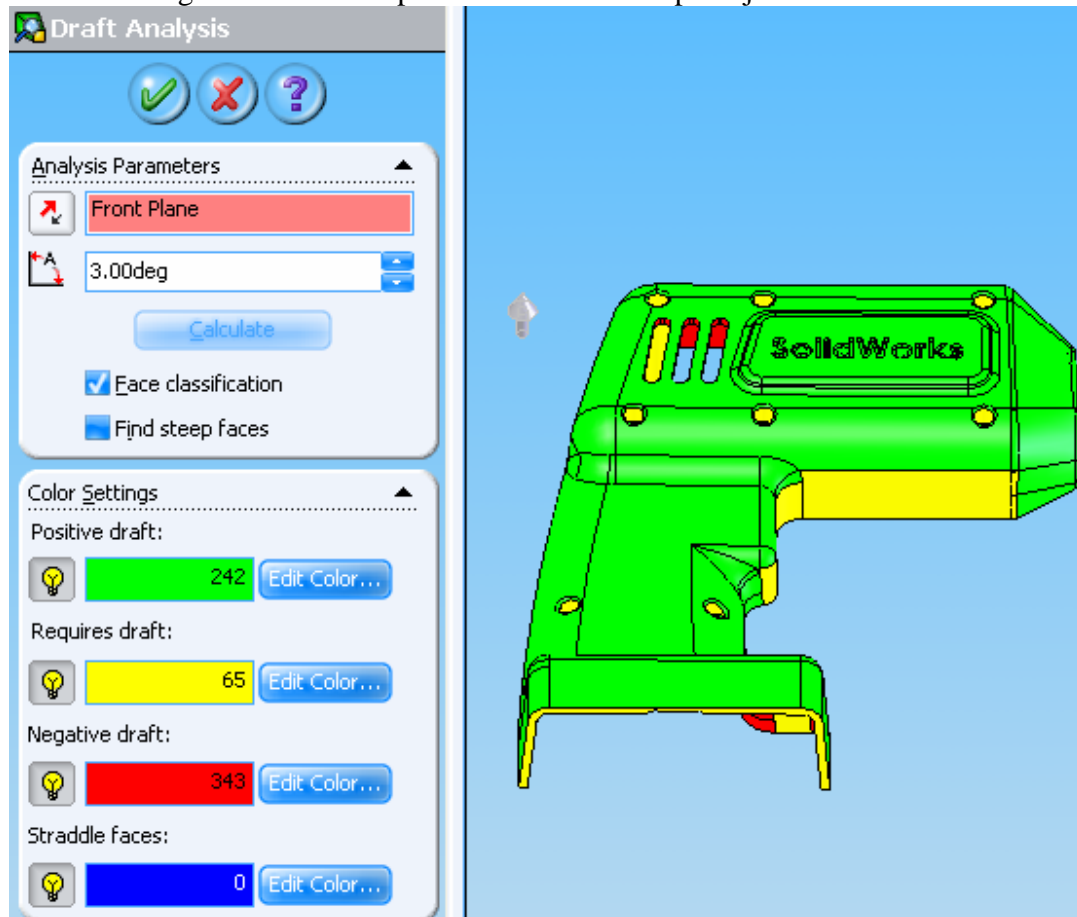
3. When file with other type, like *.igs, *.stp, etc., is imported, data translation errors may occur and the imported body or surface body will contain gaps and errors. The Solidworks application has tools to help find and repair these problematic areas on translated models as shown below.



4. 3DQuickMold advises the user to start project with new SolidWorks part.

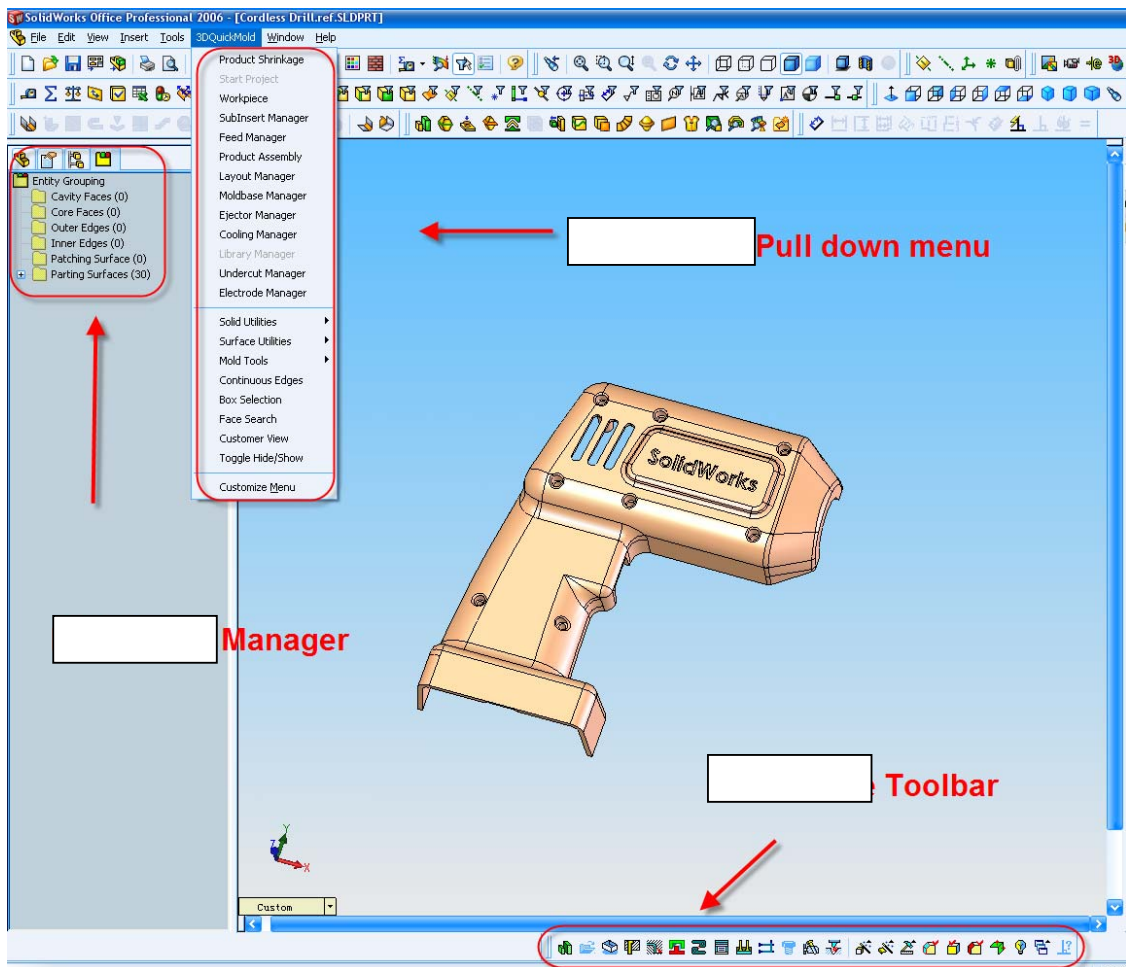
To do this, you can open a new part, insert the original plastic model file. This can retain the completeness and independency of data of the original part. Adjust a suitable origin and position. The default **mould open direction** of 3DQuickMold is the Z-axis.

5. The Solidworks application Draft Analysis  and Undercut detection  can be used to analyze the draft on a molded part. When a part is not drafted properly, the mold designer must fix the part to ensure that the part ejects from the mold.



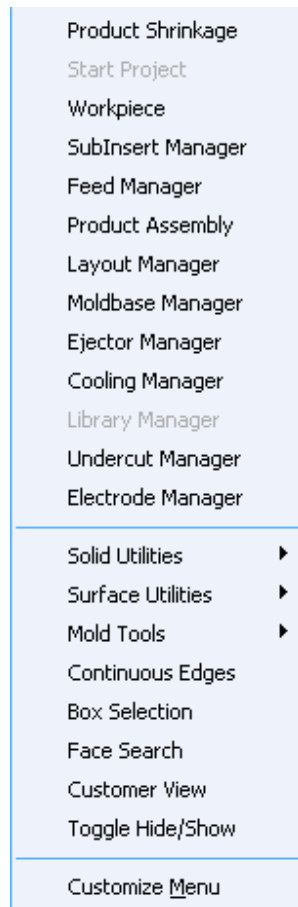
B. Software Interface

When 3DQuickMold is started, the pull down menu, toolbar and 3DQuickMold manager appear as shown below. 3DQuickMold manager appears only when a plastic part is activated, it is used to record and process geometric information relative to core/cavity separation like parting line, parting surface and Patching surface, etc..



1. 3DQuickMold pull down menu

Click 3DQuickMold in the menu bar, pull down menu pop out as follows:



each mold design module is listed below:

- Product Shrinkage: Scale the part to allow for shrinkage
 - Start Project: Start a new project
 - Workpiece: Dimension the workpiece (mold)
 - SubInsert Manager: Sub-insert design
 - Feed Manager: Runner and Gate design
 - Product Assembly: Split the core/cavity
 - Layout Manager: Arrange the core/cavity layout
 - Moldbase Manager: Load and edit mold base
 - Ejector Manager: Design ejectors
 - Cooling Manager: Cooling channel
 - Library Manager: Standard libraries for mold design
 - Undercut Manager: Slide and lifter design
 - Electrode Manager: Electrode design
-
- Solid Utilities: Utilities handling solid-related operations
 - Surface Utilities: Utilities handling surface-related operations

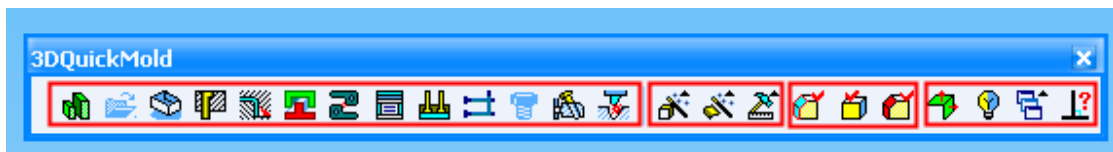
- Mold Tools : Effective tools for mold design
- Continuous Edges: Edge searching tool
- Box Selection: Face searching tool
- Face Search: Face searching tool
- Customer View: Change design view
- Toggle Hide/Show: Toggle hide/show solid bodies

2. 3DQuickMold toolbar

The corresponding icons of the above modules are listed below:

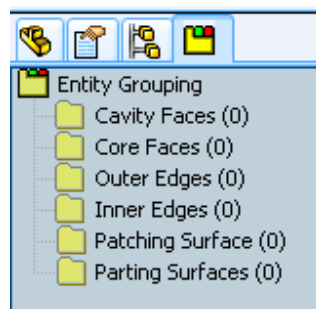


The icons list can be classified as four categories, they are Basic Functions, Useful tools, Quick selection tools and Assistant tools.



3. 3DQuickMold Manager

As shown below:



Cavity Faces : Faces belong to the cavity side

Core Faces: Faces belong to the core side

Outer Edges: Outer parting lines related to the parting surfaces

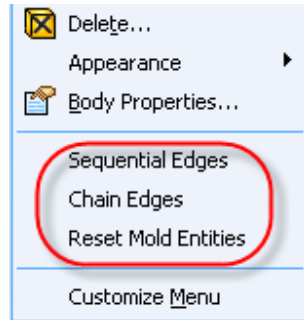
Inner Edges: Inner parting lines related to the patching surfaces

Patching Surface: Surface used to patch the holes

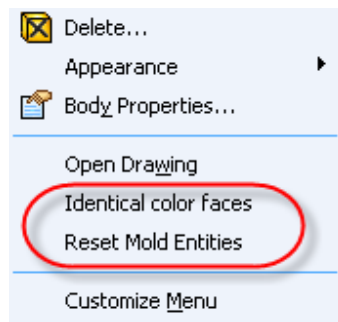
Parting Surfaces:

4. Others

Select one edge on part model and right click, pull down menu pop out as follows:



Select one face and right click, pull down menu pop out as follows:




Note that the circled function blocks activate only when the plastic part is activated.

C. Functions

Chapter 1. Design Modules and Tools

1. Product Shrinkage

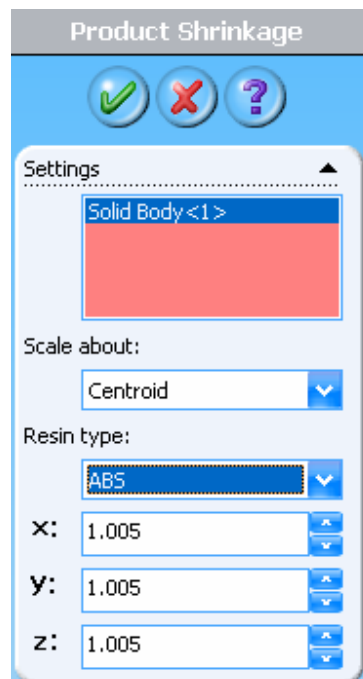
Every plastic material has a shrinkage factor assigned to it, the part has to be enlarged proportionally to compensate the contraction. After the above preparation, set the shrinkage factor of the product.

Select  on the 3DQuickMold Toolbar, for the shrinkage factor is set or not yet set, two different interfaces appear.

- Add shrinkage (mode 1)
- Edit shrinkage (mode 2)

1. Add shrinkage

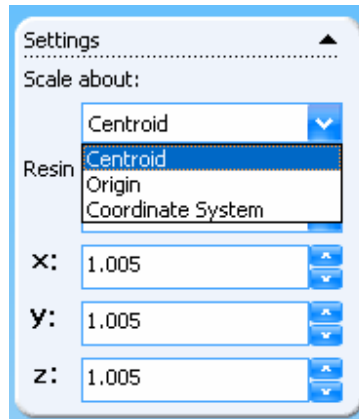
If the shrinkage factor is not defined, mode 1 of “Product Shrinkage” pops out.



Different scale factor can be set to the product.

The product can be scaled in custom x, y, z direction.

The product can be scaled about the Centroid, the Origin, or a coordinate system for shrinkage origin.



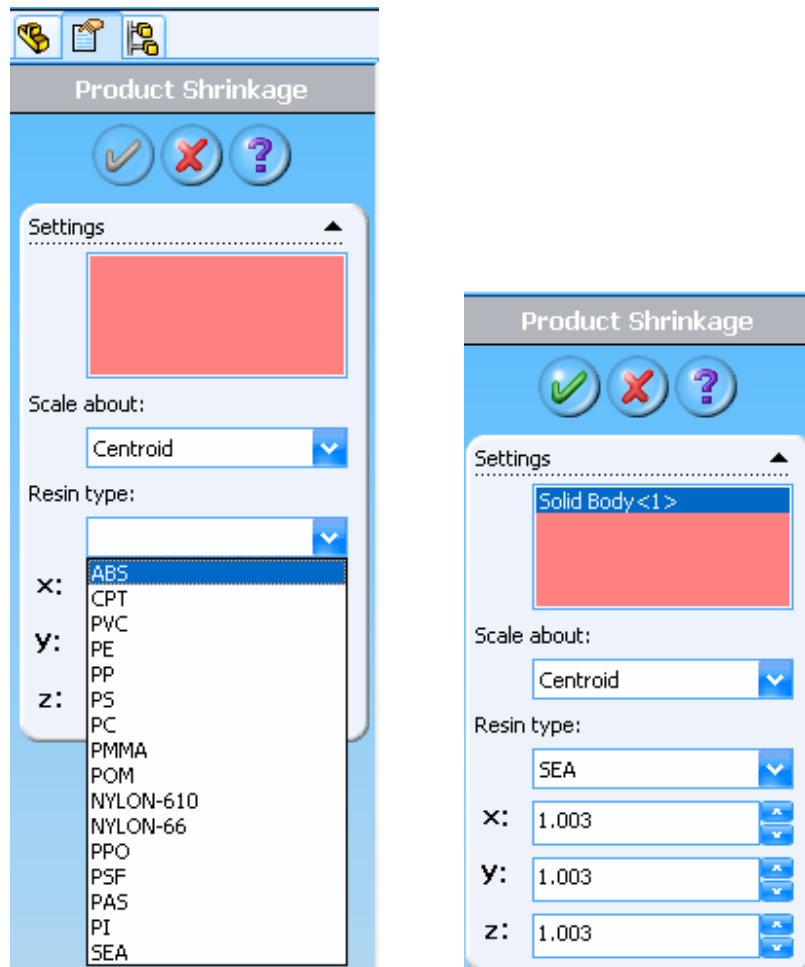
Different plastic material is available under Resin type.

The material database is located at the installation folder\ res \ shrinkage factor.xls and it can be edited.

Below shows the adding of a new material in the shrinkage factor.xls which is highlighted in grey.

D17		▼	=
	A	B	C
1		Shrinkage	Density
2	ABS	1.005	1.05
3	CPT	1.006	1.4
4	PVC	1.003	1.38
5	PE	1.016	0.95
6	PP	1.016	0.91
7	PS	1.005	1.05
8	PC	1.006	1.2
9	PMMA	1.005	1.18
10	POM	1.02	1.42
11	NYLON-610	1.016	1.1
12	NYLON-66	1.016	1.15
13	PPO	1.008	1.07
14	PSF	1.006	1.24
15	PAS	1.008	1.36
16	PI	1.008	1.36
17	SEA	1.003	1

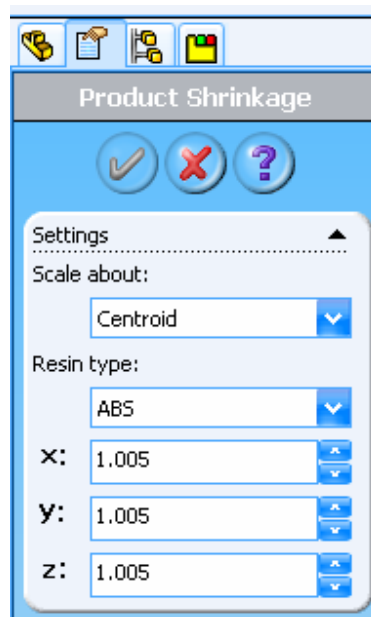
On the “Product Shrinkage” dialog, under resin type, the newly added resin can be found.



Tips: For convenience, the name of resin can be changed to the format “resin (factor)”, like  ABS(1.005)

2. Edit shrinkage

If the shrinkage is already defined, mode 2 of “Product Shrinkage” pops out.




2. Start Project

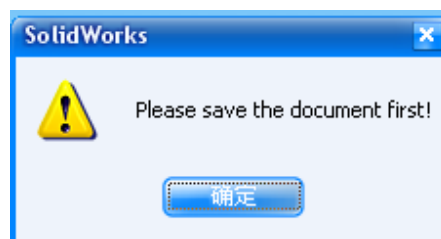
Initially, many icons on the 3DQuickMold toolbar appear to be grey which are disabled. Click Start Project to activate the icons and the 3DQuickMold Manager will be appeared as consequence.

Start Project is used for the following purposes

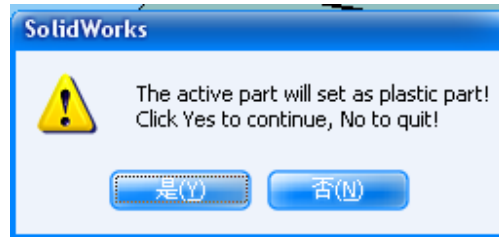
- If the plastic part is not set before, this function will set the current part as plastic part.
- If plastic part is set already, the plastic part will be activated.

3DQuickMold manager will appear, most icons on the toolbar will be activated as well.

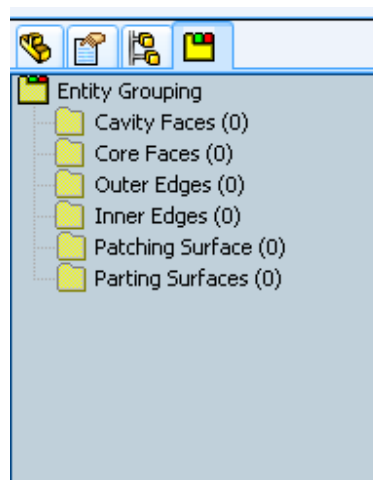
Click Start Project , if the part is not saved, the following dialog will appear . Just save the current part.



If the current model is a part file and saved, 3DQuickMold will remind you that the existing part will be considered to be the plastic part.

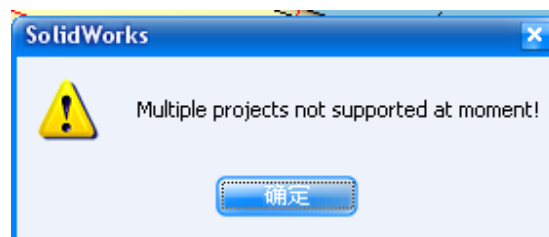


Click Yes, 3DQuickMold manager will be appeared.




Start Project button will be at the same time became grey (disabled), this is because in **3DQuickMold**, only one plastic part can be exist; only after all the documents were closed or existed from current design. Start Project button will become highlighted again.

If two part set as plastic part before exist at the same time, the following dialogue box pops up.

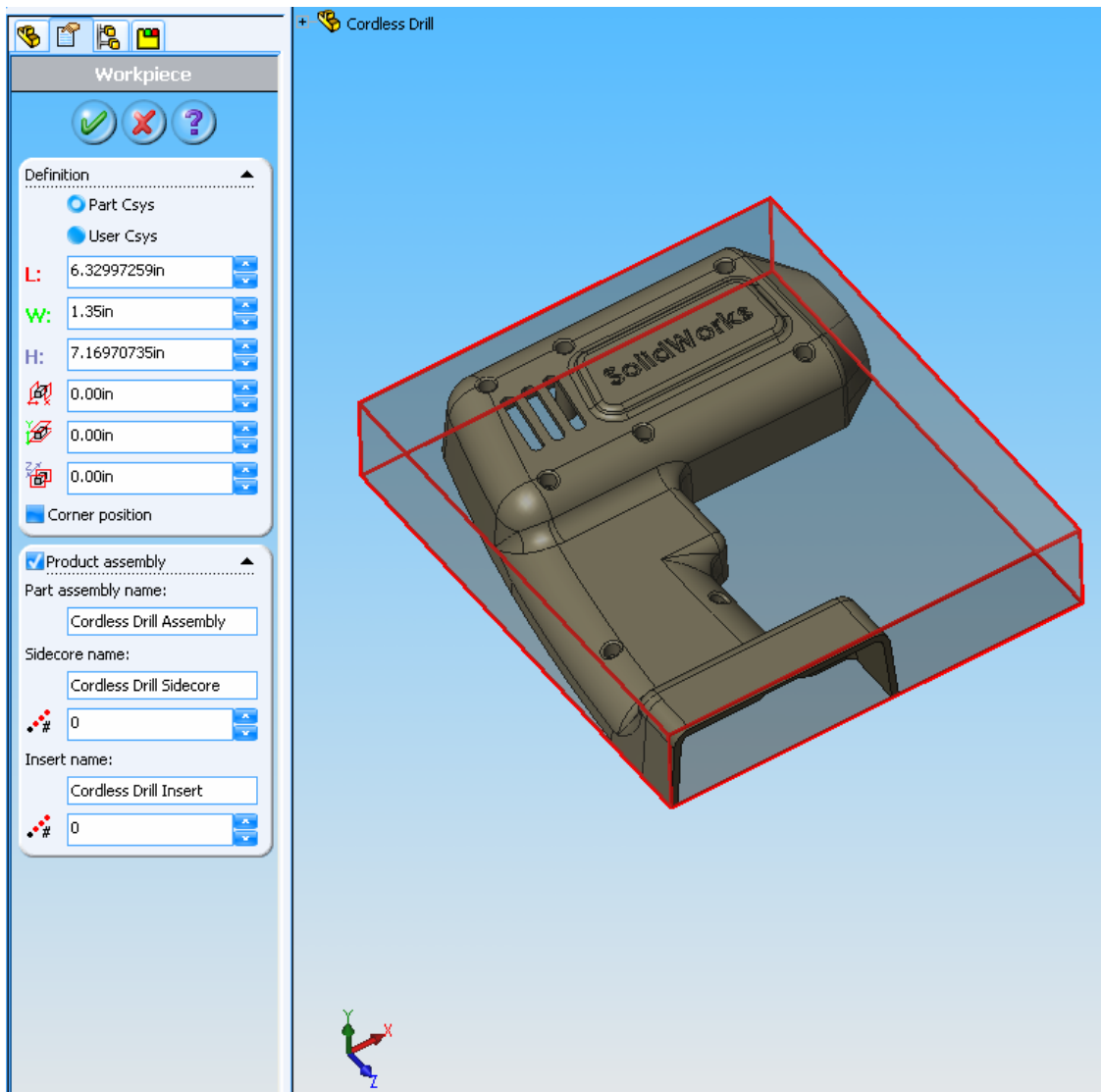


3. Workpiece

Workpiece is used to define the dimension of mold core. It should be used before building the parting surface.

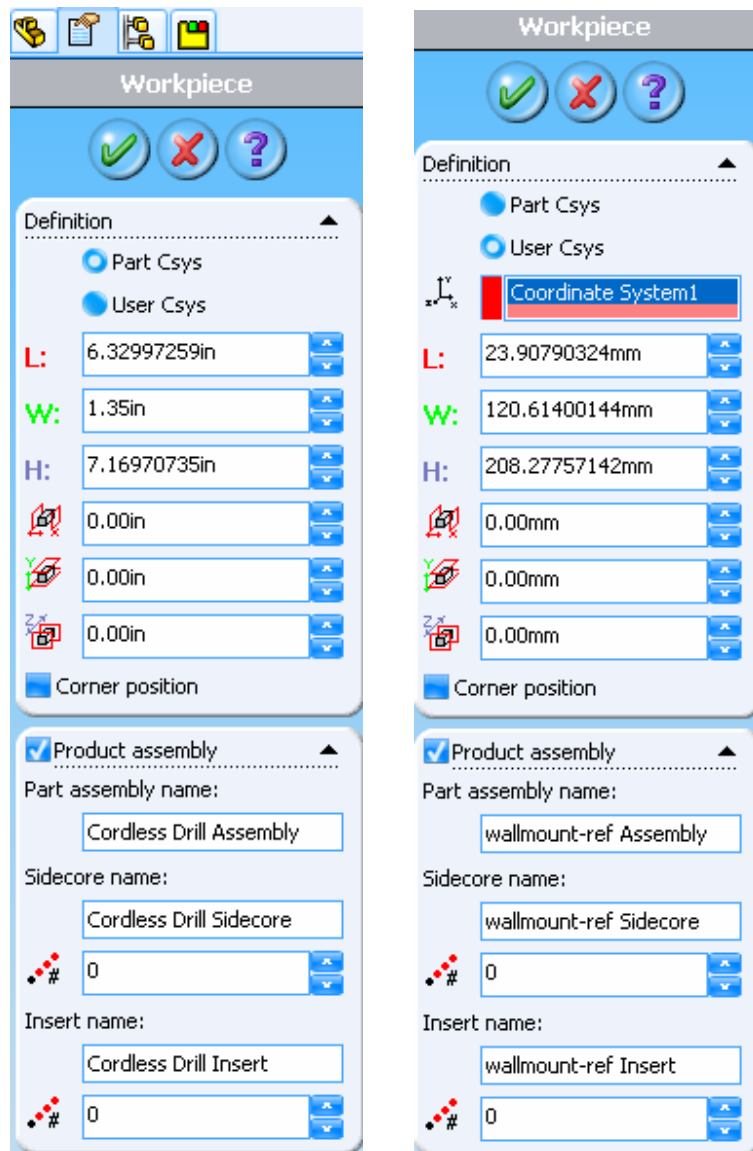
Click Workpiece  on the 3DQuickMold Toolbar, 3DQuickMold will define the smallest dimension of the workpiece according to the default coordinate system and create a preview.

Adjust the size when needed.



Here provides two coordinate systems to define workpiece.

- Part Csys: Use the part coordinate system as the reference.
- User Csys: Use the user-defined coordinate system as the reference.



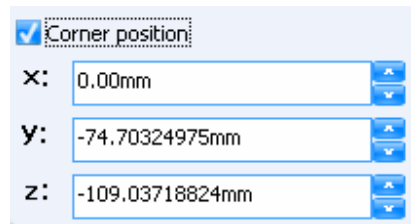
In Definition, define the dimension of workpiece and the offset of the whole workpiece along the x, y, and z direction.

- Length: Define the length of the workpiece
- W:** Width: Define the width of the workpiece
- H:** Height: Define the height of the workpiece
- : Shift the workpiece along the x-direction
- : Shift the workpiece along the y-direction
- : Shift the workpiece along the z-direction

Check Corner Position, the position of the workpiece relative to the coordinate system is shown.

The value in Corner position is the minimum value of the workpiece in the x, y, z direction.

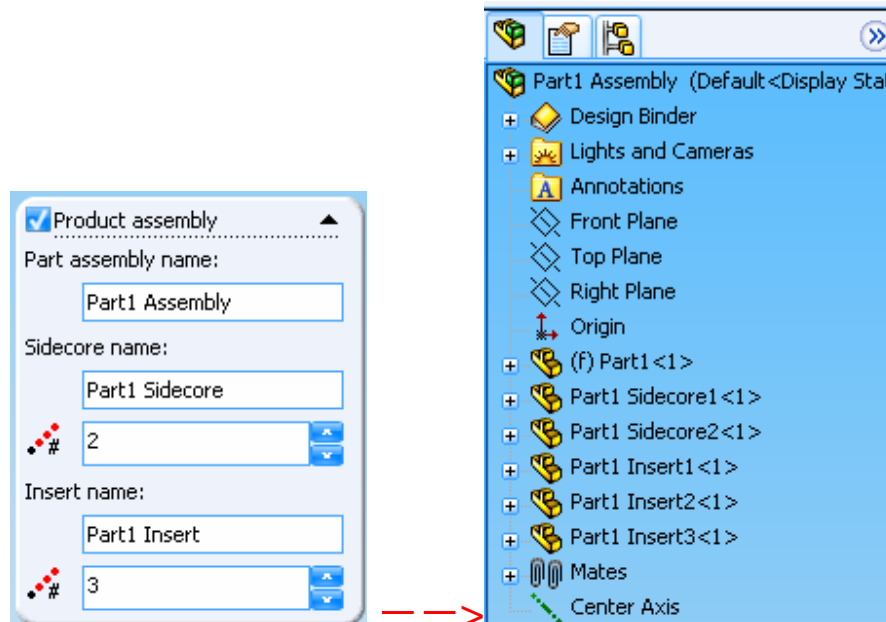
The workpiece can be offset by changing the value of x, y, z.



Check Product assembly, 3DQuickMold will create a * Assembly.sldasm assembly file (* is the name of the plastic part).

The number of Sidecore and Insert can be defined here. It can be defined in the Product Assembly which will be mentioned later.

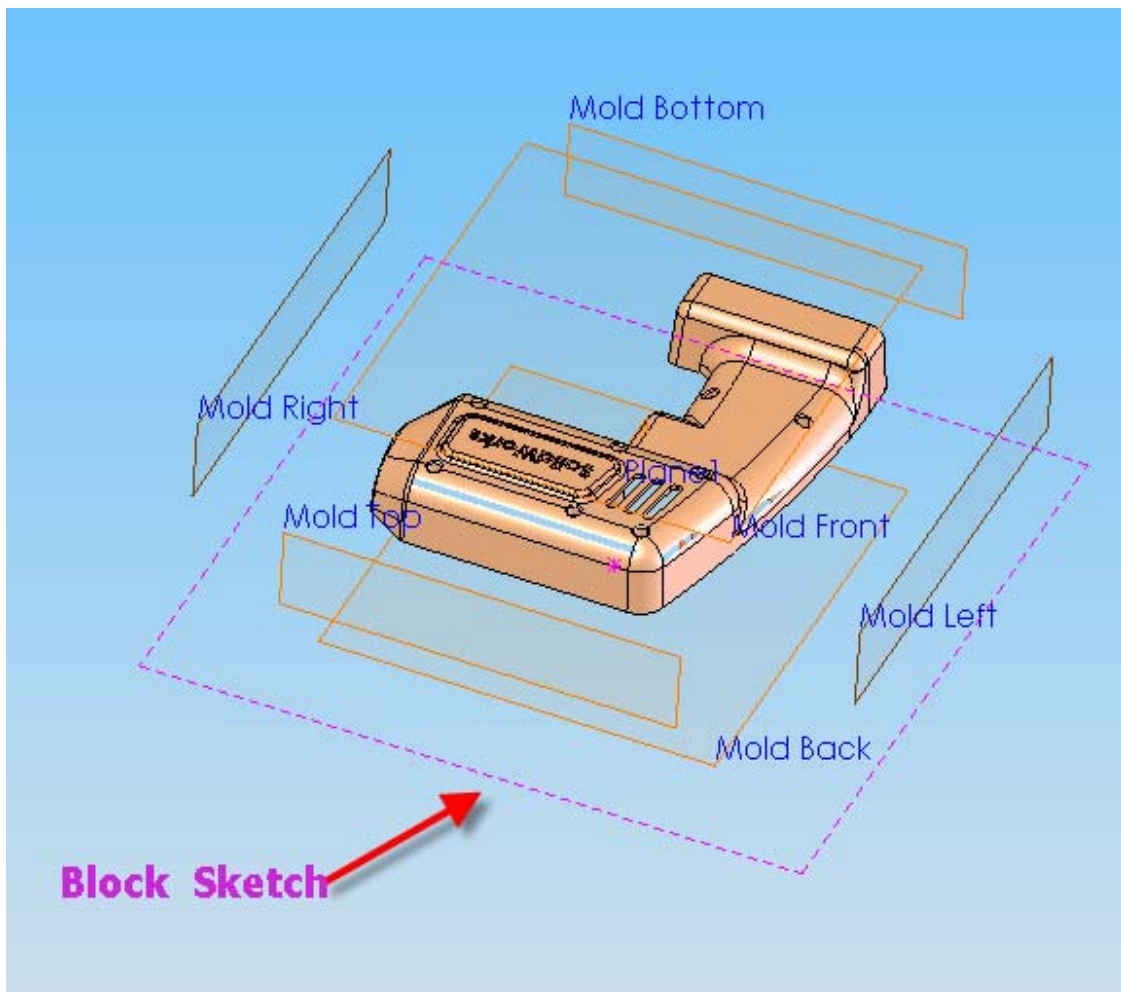
After OK, remember to save the created * Assembly.sldasm file, otherwise it will be a blank file if Solidworks is ended.



After setting is completed, click OK.

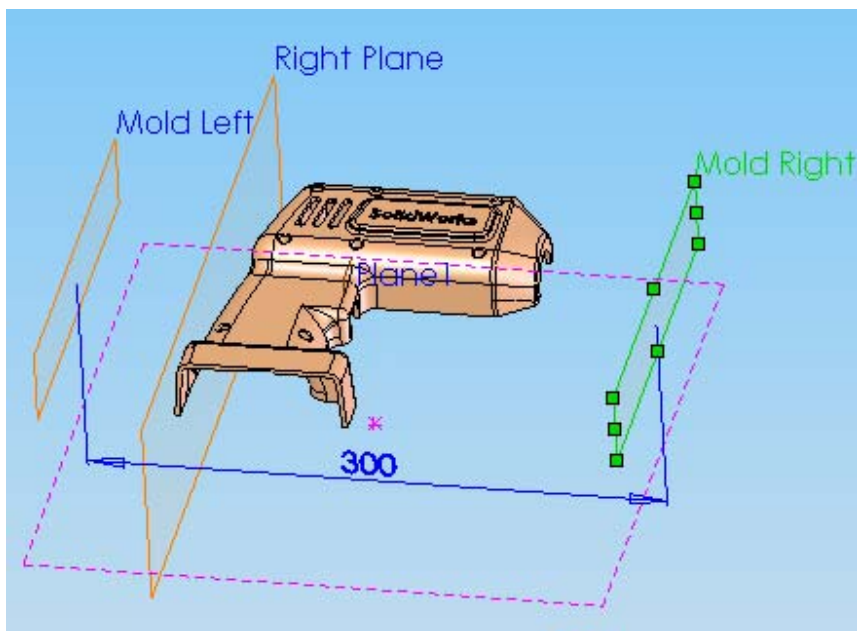
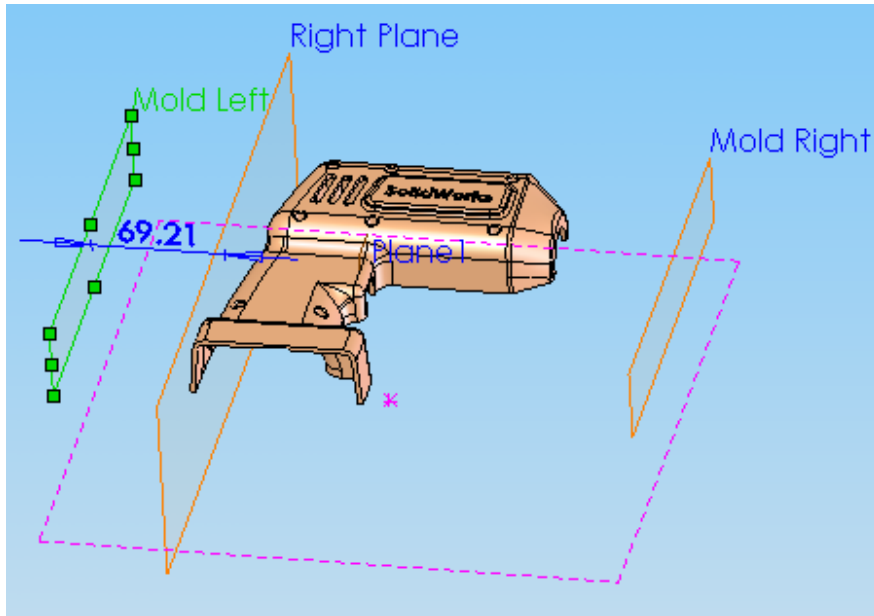
6 reference planes with the workpiece as border are created. (the first five reference planes are hidden, Mold Front is shown; the distance between Mold Front and sketch is the height of the workpiece, the size of the whole work piece is visualized.

A sketch of Block Sketch with Mold Back as reference is also created, This block sketch determine the size of the final size of the core and cavity, the dimension of the sketch is the dimension of the projected image of the workpiece on the xy plane.

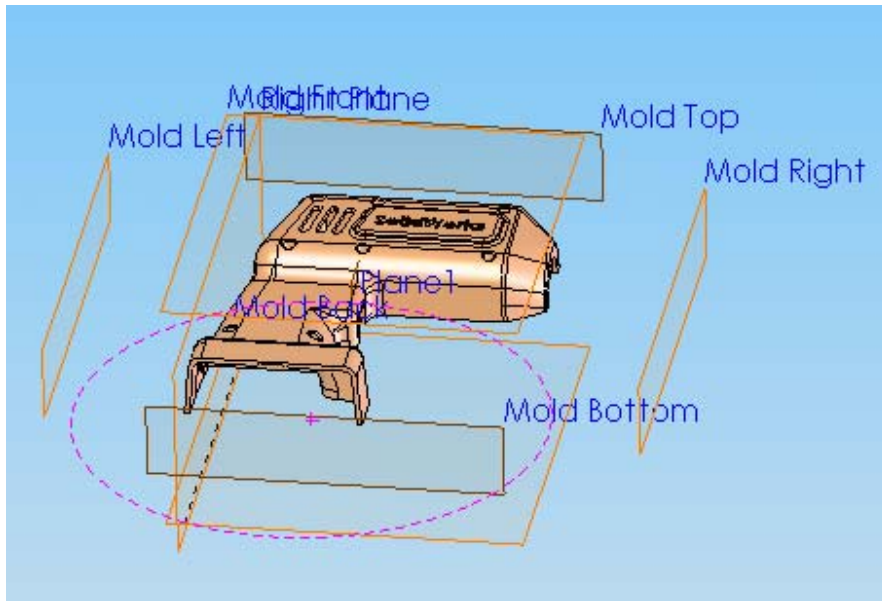


- Mold Left ,Mold Bottom ,Mold Back, are created with reference to the starting reference planes, which is the Right Plane, Top Plane, Front Plane of the part.
- Mold Right, Mold Top, Mold Front are created with reference to Mold Left ,Mold Bottom ,Mold Back.

As the following pictures shown.



The sketch of the Block Sketch is the bottom shape of the Cavity and Core created later. The sketch can be edited in the edit feature.



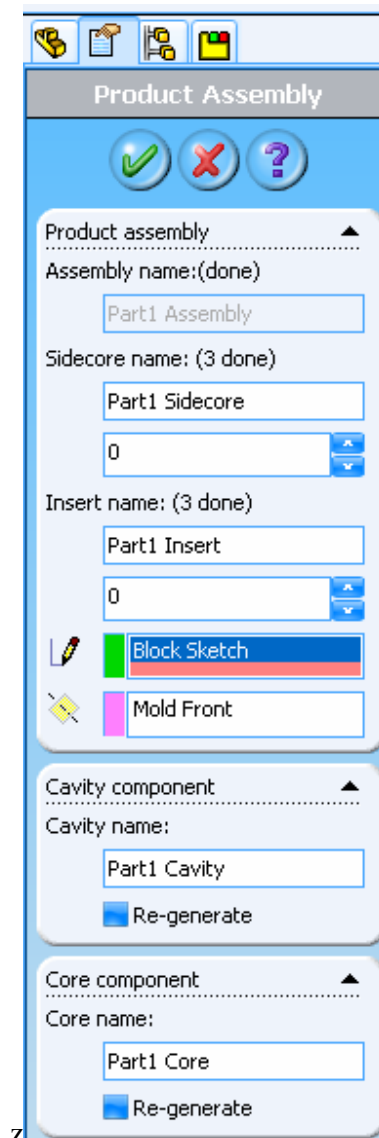
4. Product Assembly

This function is mainly used:

- as the final stage of the automatic split core/cavity
- to regenerate the core/cavity when the parting surface is changed.
- to add additional side cores or sub-insert parts to the product assembly

Click  on the 3DQuickMold Toolbar, the property manager pop out.

If the number of Sidecore and Insert of the workpiece is not specified before, specify it in the Product assembly.



Product Assembly

Product assembly ▲

Assembly name: (done)

Part1 Assembly

Sidecore name: (3 done)

Part1 Sidecore

0

Insert name: (3 done)

Part1 Insert

0

Block Sketch

Mold Front

Cavity component ▲

Cavity name:

Part1 Cavity

Re-generate

Core component ▲

Core name:

Part1 Core

Re-generate

Assembly name: Input name of the product assembly. If that assembly is already generated in Workpiece, the name field will be greyed out.

Sidecore name: Input the name of the side core, the number inside the bracket indicate the existing number of sidecore. If the number of sidecore is not sufficient, add by selecting a suitable number;

Insert name: The configuration of Insert is the same as sidecore

Sketch: Select sketch of the workpiece, **3DQuickMold** will select the sketch of Block Sketch generated in workpiece.

Plane: **3DQuickMold** will automatically select the Mold Front generated in Workpiece as the reference plane; This plane will be used to define the end condition of extrude feature.

Custom sketch and plane can also be used. It is recommended to use the default selection.

Cavity name: Input the name of the cavity;

Re-generate: If the property of the finished cavity is changed, rebuild to replace the original cavity.

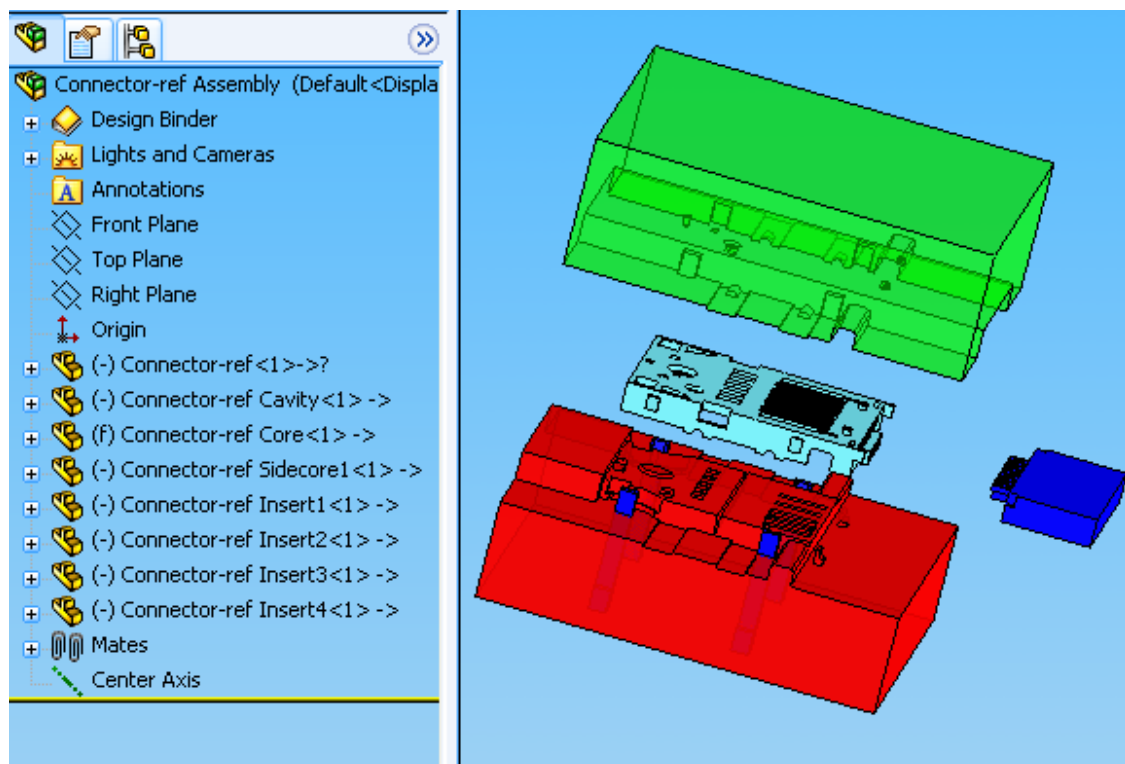
Core name: Input the name of the core;

Re-generate: if the property of the finished core is changed, rebuild to replace the original core.

Tips: After Re-generate, the newly generated part will replace the previous core/cavity, any special feature on the previous core/cavity will not pass to the new one.

After the setting is completed, click OK

3DQuickMold will generate * core.sldprt, * cavity.sldprt and re-build * assembly.sldasm, * side core1.sldprt * side core2.sldprt, ...



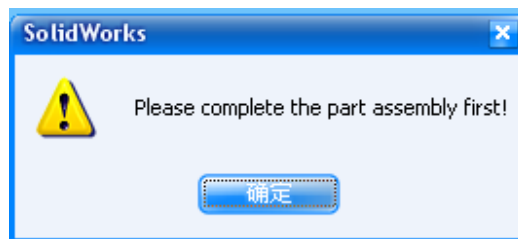
5. Layout Manager


The functions of Layout Manager

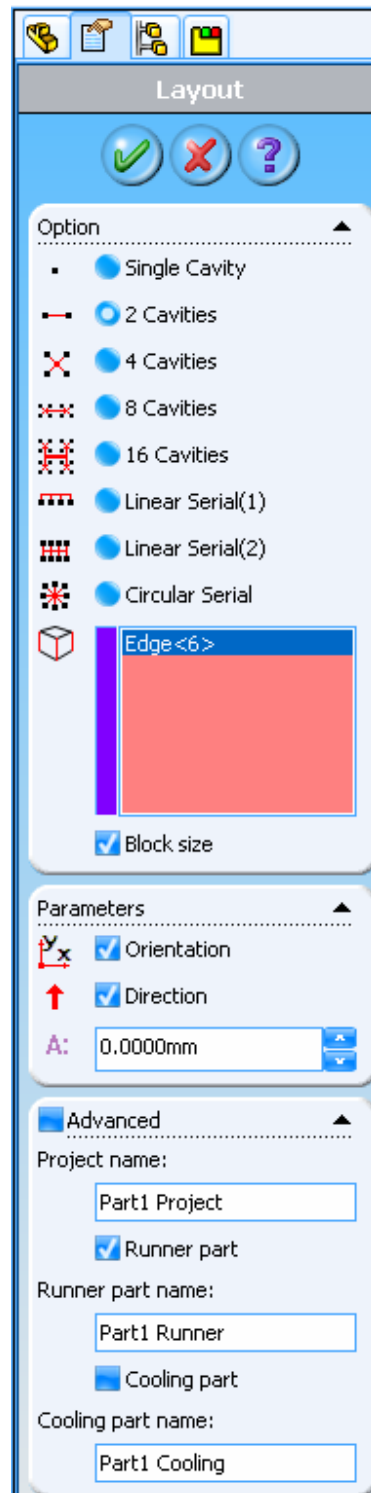
1. It can arrange the position of cavities of a multi-cavity mold, a * Project.sldasm(top assembly file) is built. This function can be used after the * Assembly.sldasm(product assembly) is built. To enable this function, the plastic part must be activated.
2. Edit the number and position of cavities of a* Project.sldasm(top assembly file) with finished layout.

Note: for single-cavity mold, it also requires Layout, otherwise the * Project.sldasm cannot be generated.

If there is not any * Assembly.sldasm file, a warning message will pop out.



Click  on the 3DQuickMold Toolbar, the Layout Manager pops out. Several patterns are available for selection.



Available patterns as follows:
 Single cavity, 2 cavities ,4 cavities, 6 cavities, 8 cavities, 16 cavities , Linear Serial(1),
 Linear Serial(2), Circular Serial

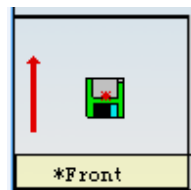
Option

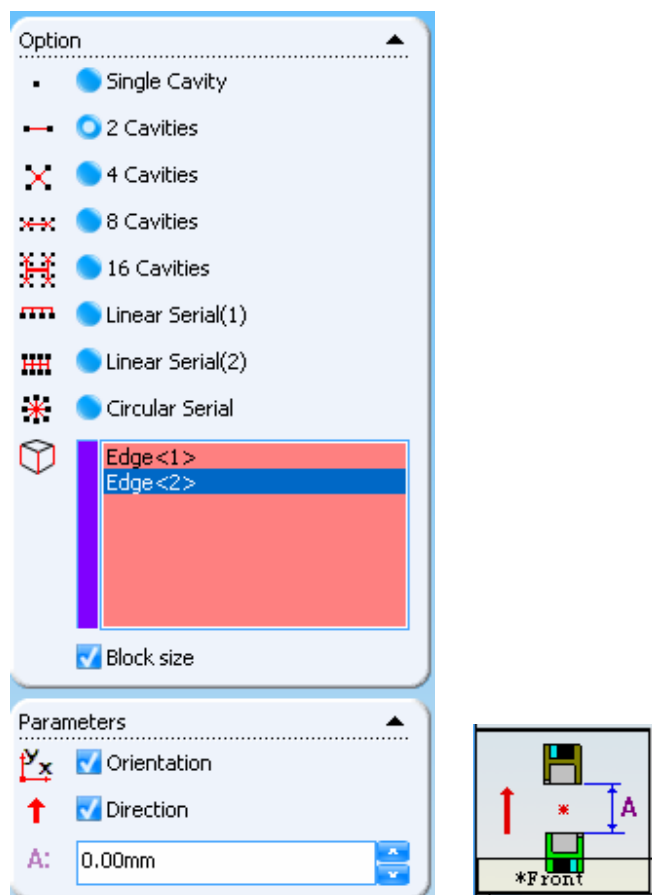
- ☒ Single Cavity
- ☐ 2 Cavities
- ☐ 4 Cavities
- ☐ 8 Cavities
- ☐ 16 Cavities
- ☐ Linear Serial(1)
- ☐ Linear Serial(2)
- ☐ Circular Serial

☒ Block size

Parameters

- ☒ Orientation
- ☒ Direction





Dimension A is used to control the pattern details.

Option

- ☐ Single Cavity
- ☐ 2 Cavities
- ☒ 4 Cavities
- ☐ 8 Cavities
- ☐ 16 Cavities
- ☐ Linear Serial(1)
- ☐ Linear Serial(2)
- ☐ Circular Serial

Edge<1>
Edge<2>

☒ Block size

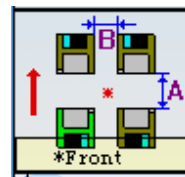
Parameters

☒ Orientation

☒ Direction

A: 0.00mm

B: 0.00mm



Dimension A and B are used to control the pattern details.

Option

- ☒ Single Cavity
- ☐ 2 Cavities
- ☐ 4 Cavities
- ☐ 8 Cavities
- ☐ 16 Cavities
- ☐ Linear Serial(1)
- ☐ Linear Serial(2)
- ☐ Circular Serial

Edge<1>
Edge<2>

☒ Block size

Parameters

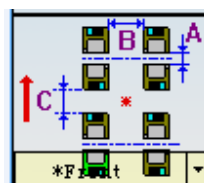
☒ Orientation

☒ Direction

A: 0.00mm

B: 0.00mm

C: 0.00mm



Option

Single Cavity

2 Cavities

4 Cavities

8 Cavities

16 Cavities

Linear Serial(1)

Linear Serial(2)

Circular Serial

Edge<1>

Edge<2>

Block size

Parameters

Orientation

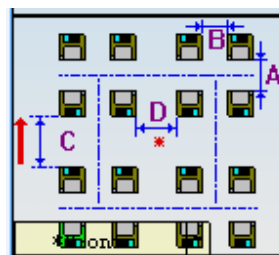
Direction

A: 0.00mm

B: 0.00mm


C: 0.00mm

D: 0.00mm



Option ▲

- ☐ Single Cavity
- ☐ 2 Cavities
- ☐ 4 Cavities
- ☐ 8 Cavities
- ☐ 16 Cavities
- ☐ Linear Serial(1)
- ☐ Linear Serial(2)
- ☐ Circular Serial




Edge<1>


Edge<2>

☒ Block size

Parameters ▲




☒ Orientation



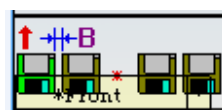
☒ Direction

B:

▲▼



▲▼



Option

- ☒ Single Cavity
- ☐ 2 Cavities
- ☐ 4 Cavities
- ☐ 8 Cavities
- ☐ 16 Cavities
- ☐ Linear Serial(1)
- ☐ Linear Serial(2)
- ☐ Circular Serial

Edge<1>
Edge<2>

☒ Block size

Parameters

☒ Orientation

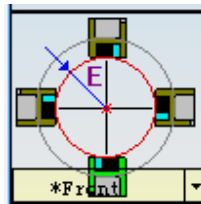
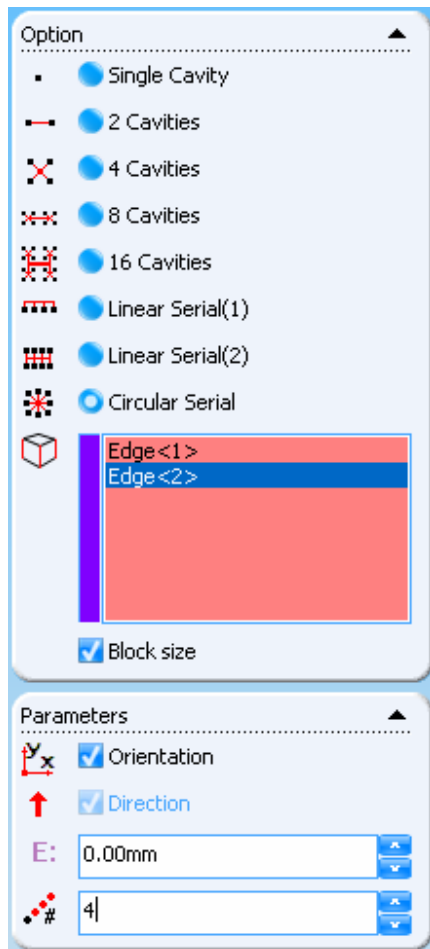
☒ Direction

A: 0.00mm

B: 0.00mm

8





Note: for the construction of 32 cavities, 64 cavities, 128 cavities mold, etc. first select 16 cavities to create the pattern. Then in Edit pattern, use Mirror Copy to finish.

Reference edges: They are used for preview purpose only.

If outer edges are defined in the **Entity Grouping/Outer Edges**, 3DQuickMold will automatically select all **Outer Edges**.

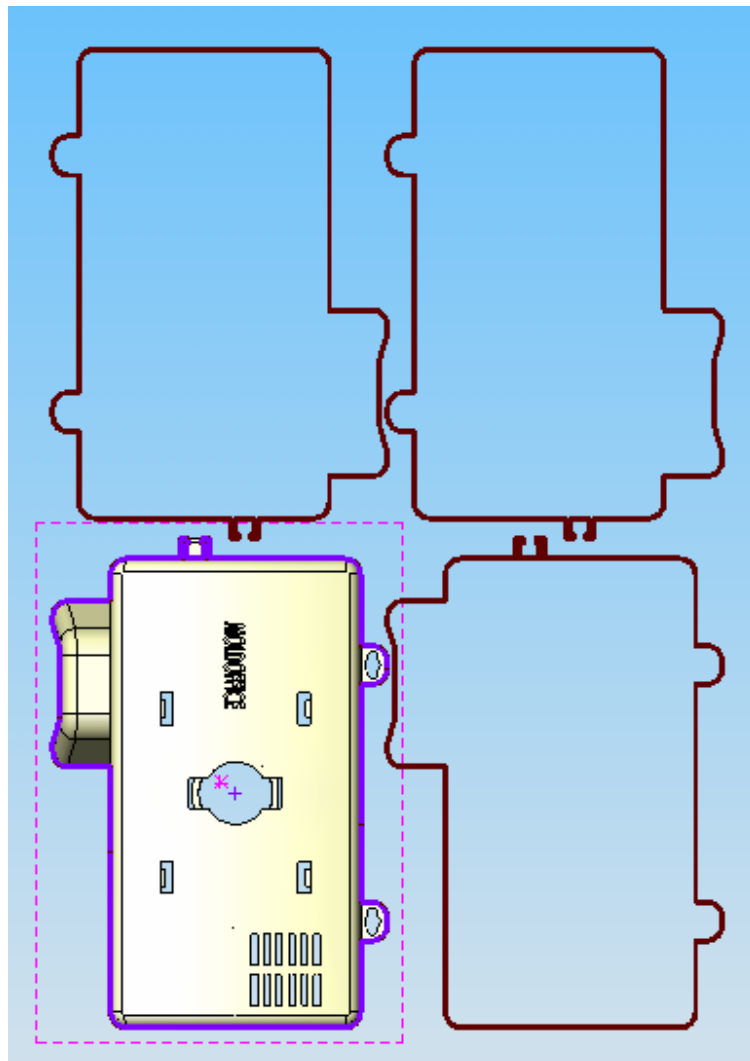
If nothing is selected in the **Entity Grouping/Outer Edges**, 3DQuickMold will select the circular edges on the part as the reference edge.

Edge can be added or deleted manually. The quantity of edge enhances the convinence of viewing the orientation of cavities inside the mold but it does not affect the cavity layout.

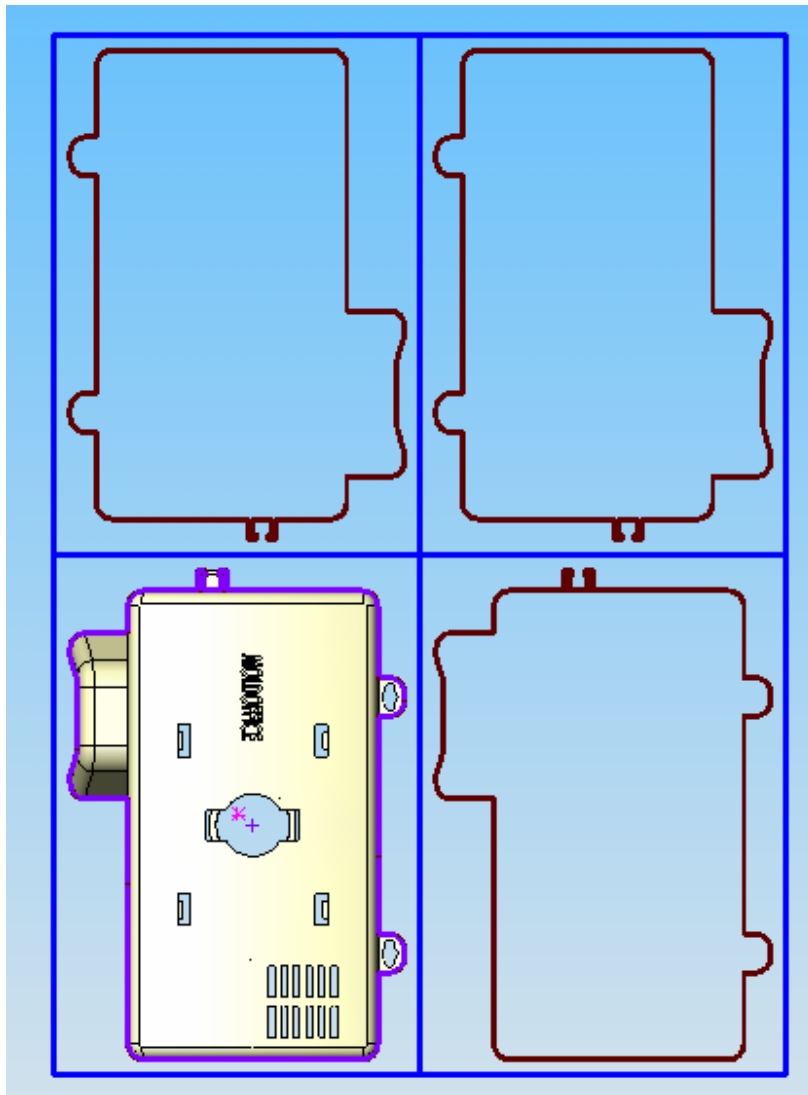
Block size: By default, the Block size is checked, maximum boundaries of core/cavity block is used to calculate the distance between each cavity.

If Block size is unchecked, maximum boundaries of the plastic part is used to calculate the distance between each cavity.

If Block size is unchecked, 0 distance between each core/cavity is preview as following

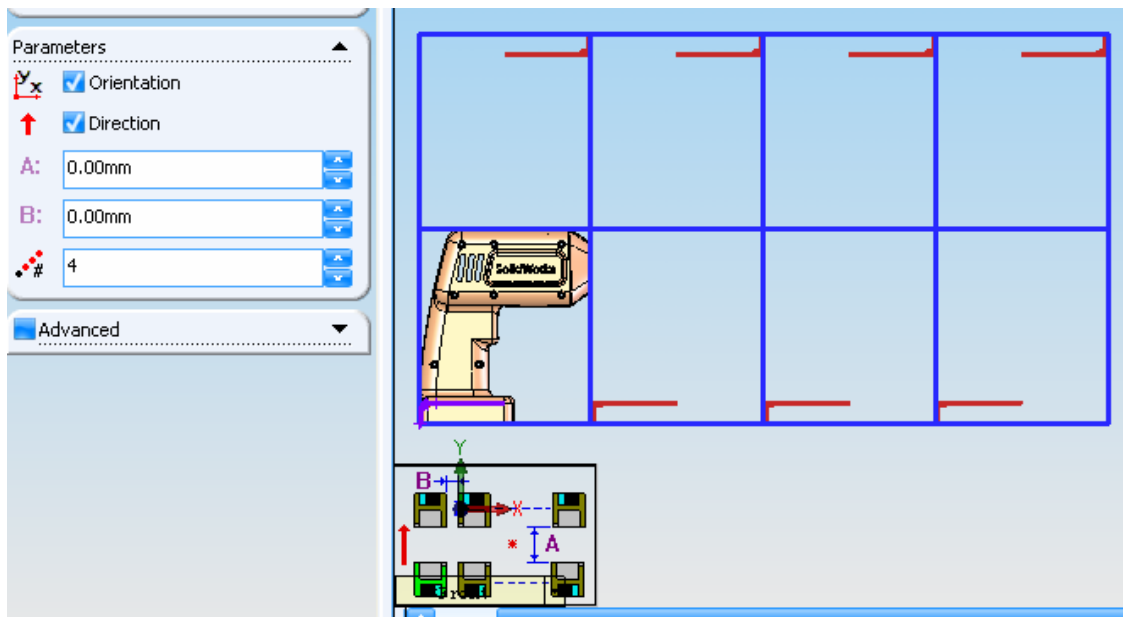


While Block size is checked and 0 distance between each core/cavity is set, the layout preview is shown as following picture.

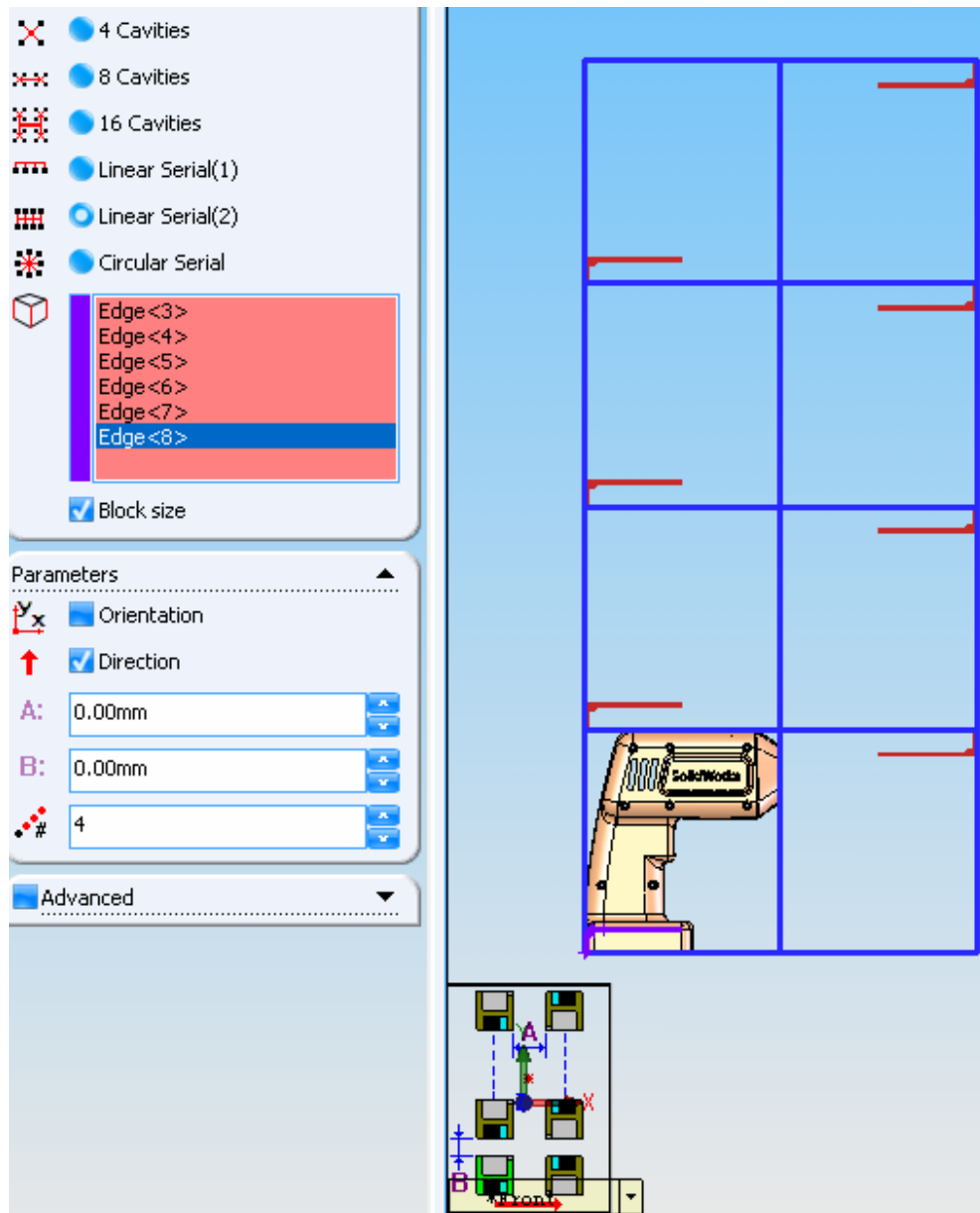


Orientation: Set the relative orientation of the other cavities to the original starting cavity

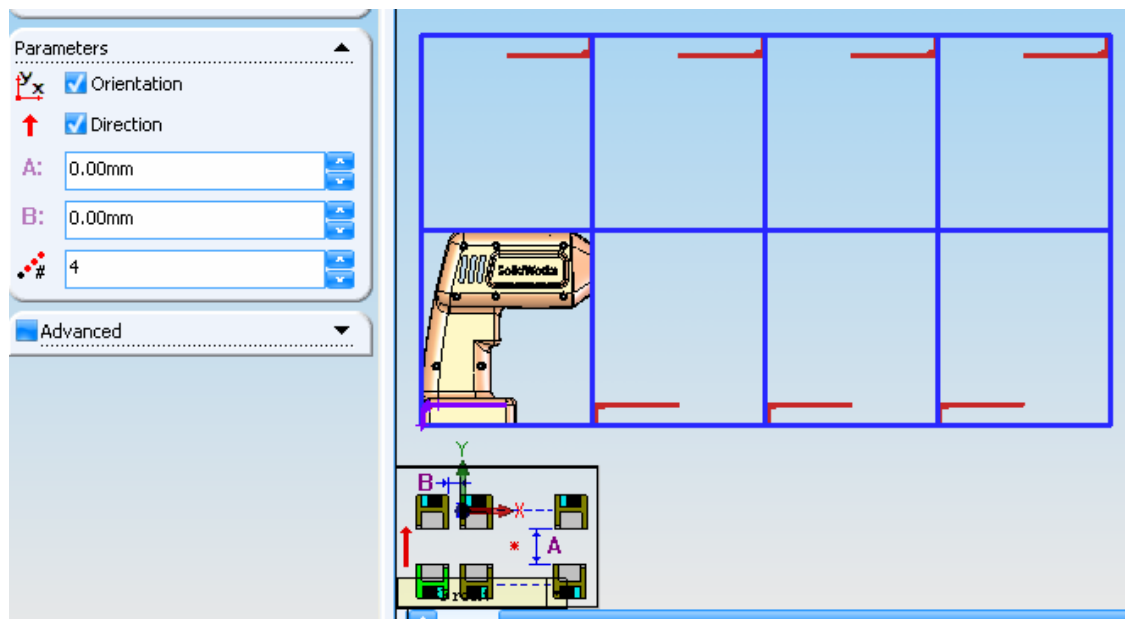
The position of sidecore and the setting of runner should be taken into consideration when the orientation and direction are changed.



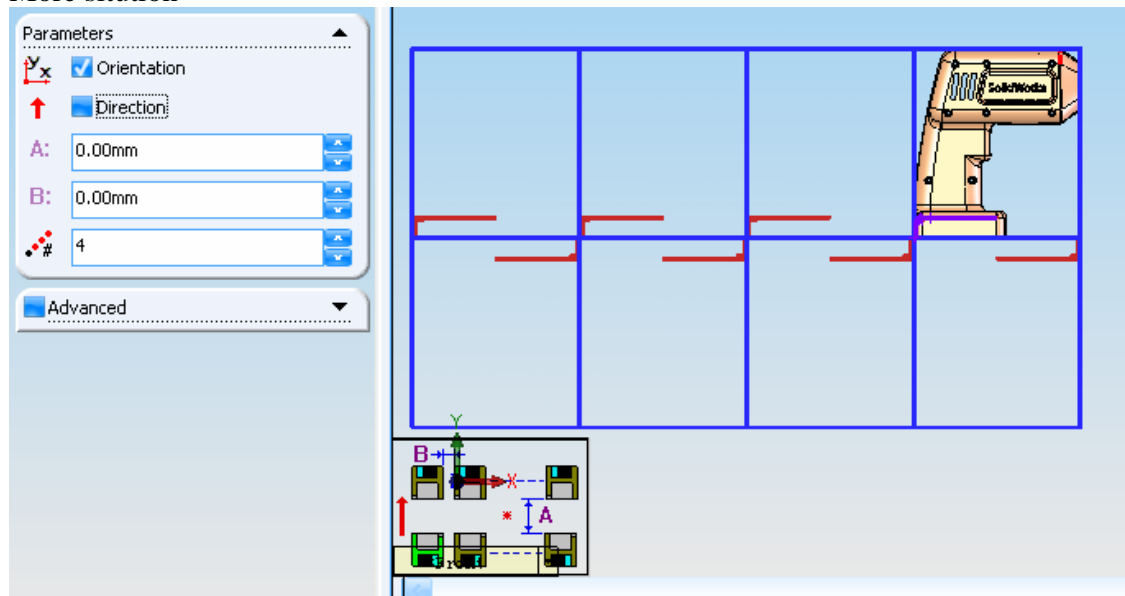
Different situation with Orientation unchecked and Direction checked



Another situation with both Orientation and Direction checked

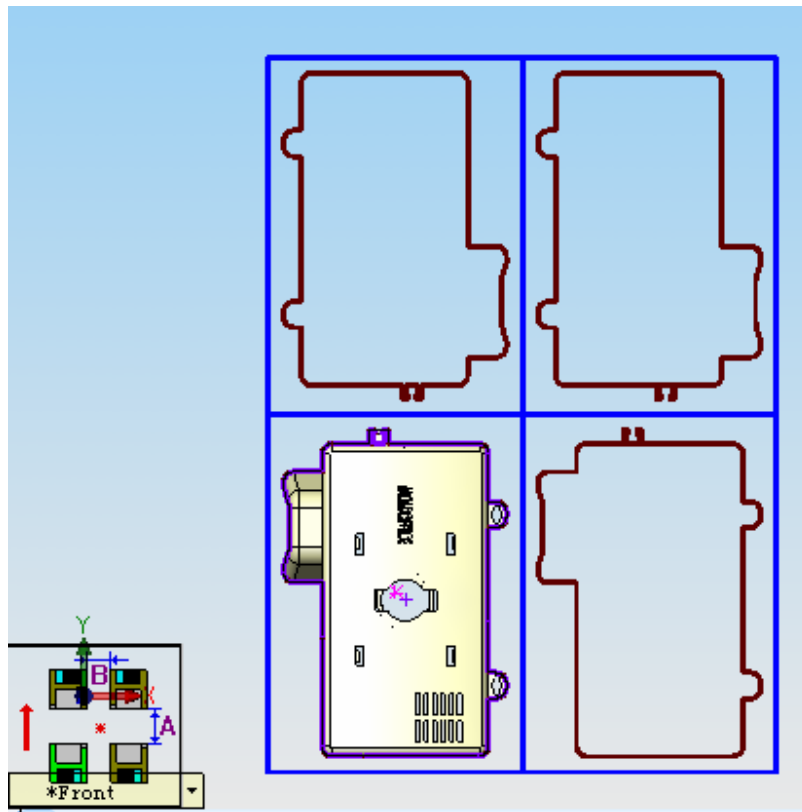



More situation



By check or uncheck **Orientation** and **Direction**, the other cavities can be positioned on any of the position in the top, down, left and right of the original cavity

A: (there may be also B:, C:, E:) Parameters to control the layout dimension

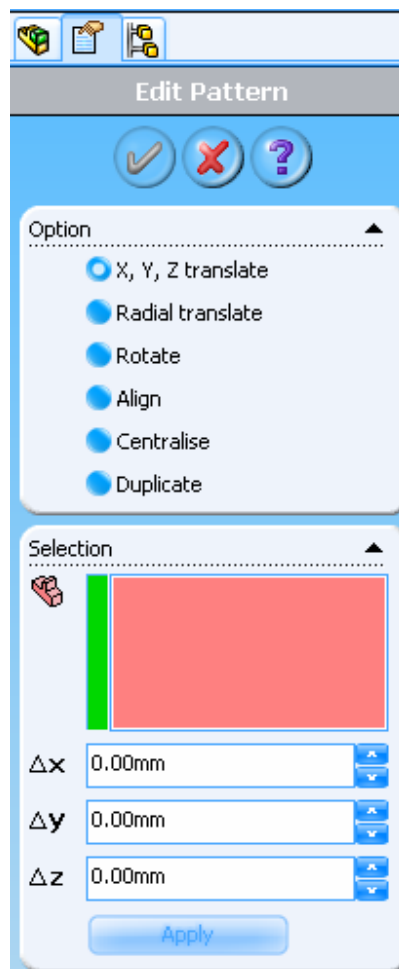


Tips: change the orientation of the preview to Front view, the original cavity will look like the icon at  the bottom left

For product which have been already **Layout**, if **Layout** is click again, the **Edit Pattern Manager** will pop out;

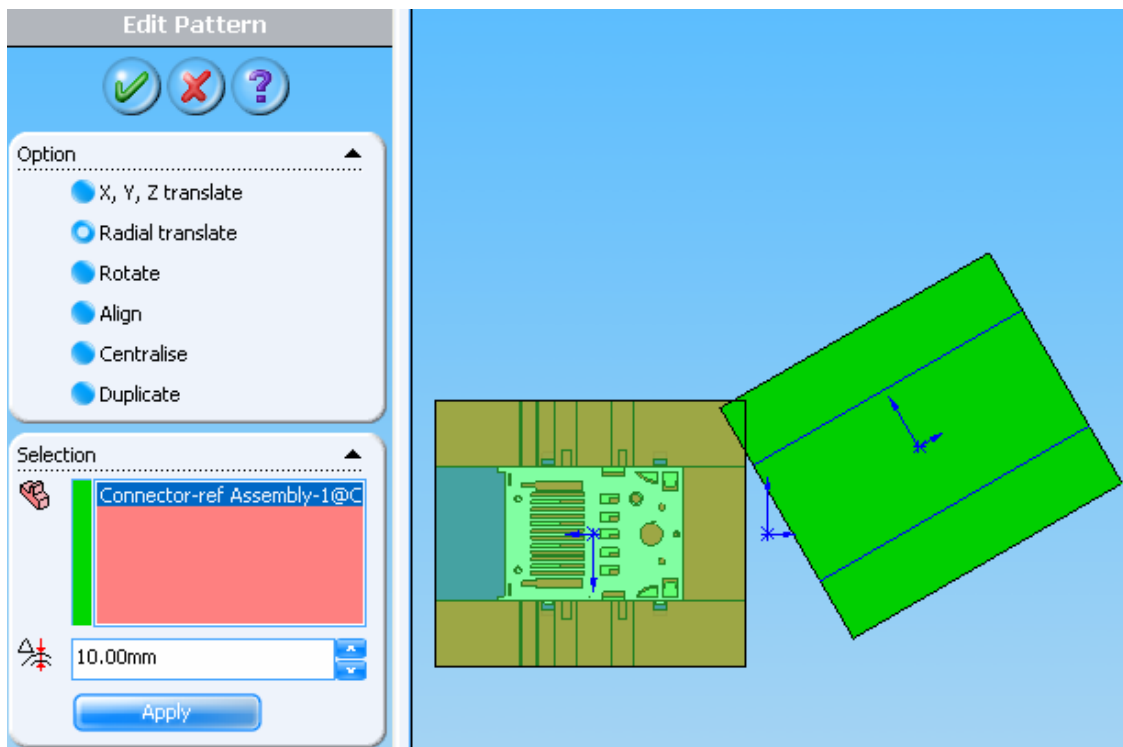
Note: The value entered in the field will return to 0 after clicking **Apply** as the value is the relative value. When the value has returned to 0, no change will appear if **Apply** is clicked. If the value entered is not suitable, enter the negative value to undo the change.

X, Y, Z translate: Translate the selected cavity along the X, Y, Z direction
After the mold base is inserted, Edit Z to alight the parting surface on the mold base.

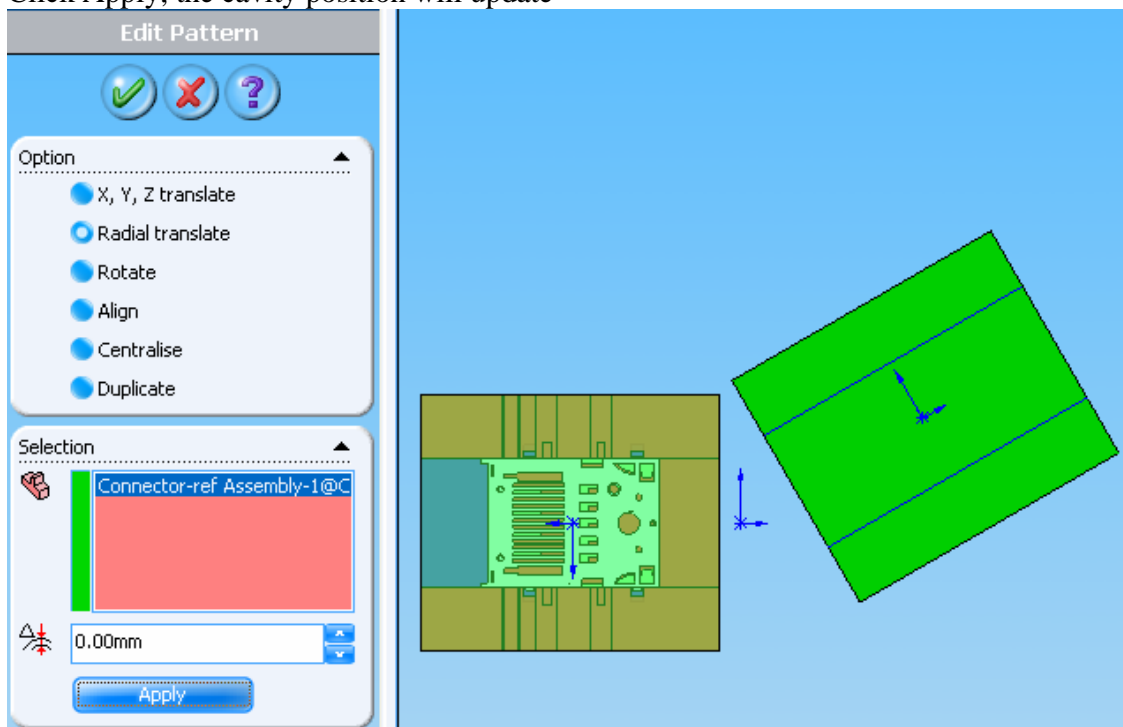


Radial translate: Move the selected cavity along the line joining the center of the selected cavity and the center of the layout.

Select the required cavity, enter the translation distance



Click Apply, the cavity position will update



Rotate: Rotate the selected cavity

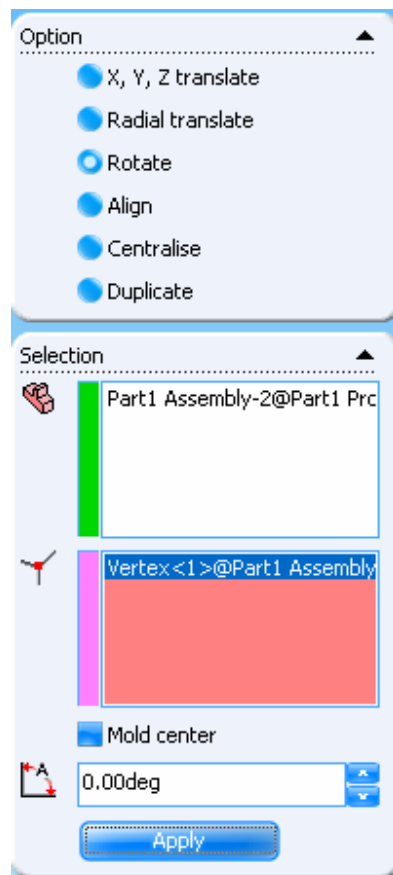
Different situations will depend on Mold Center's state.

- Mold Center is unchecked

1. Rotate the selected cavity about a selected point and perpendicular to the Z-axis.
2. If several points are selected, the rotation center will be the center of the polygon formed by the points, the cavity is rotated perpendicular to the Z-axis.
3. If no point is selected, the cavity is rotated about its center.

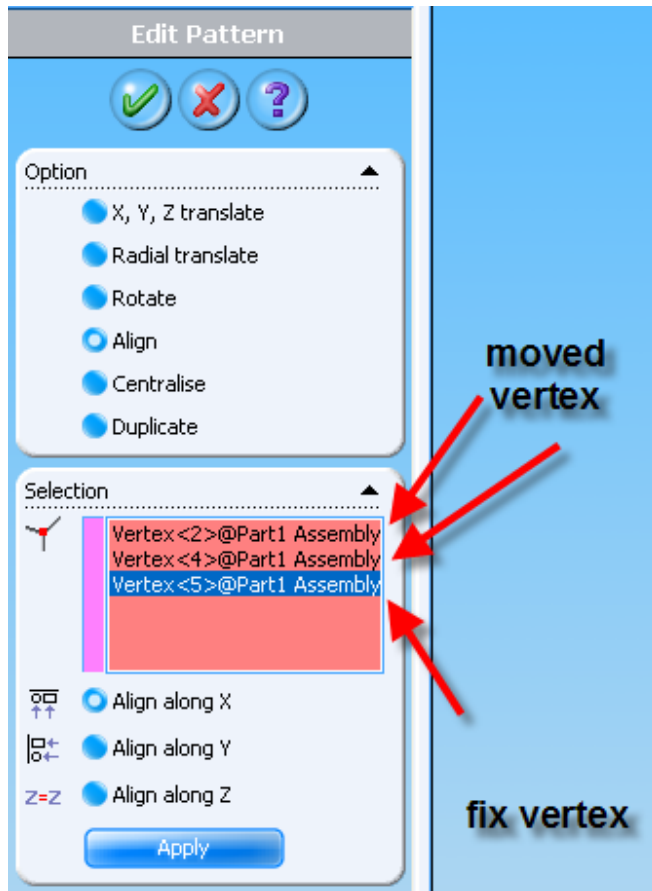
- Mold Center is checked

The cavity is rotated perpendicular to the Z-axis about the Origin of the generated * Project.sldasm

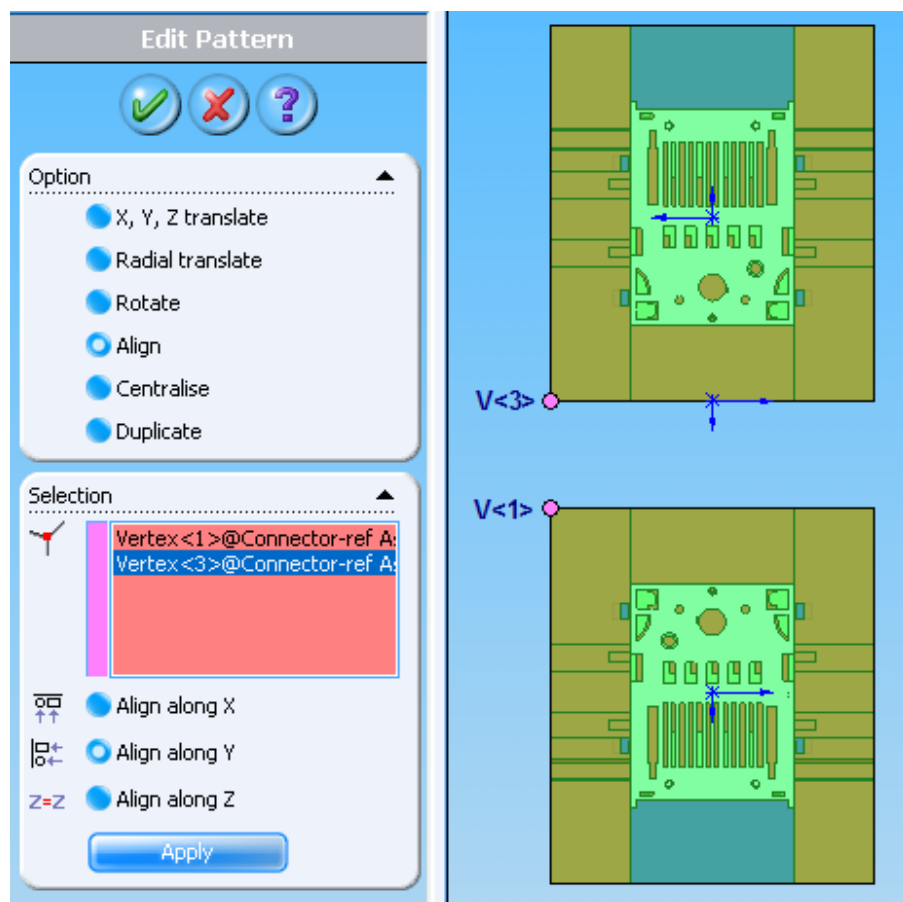


Align: align horizontally or vertically to the selected point

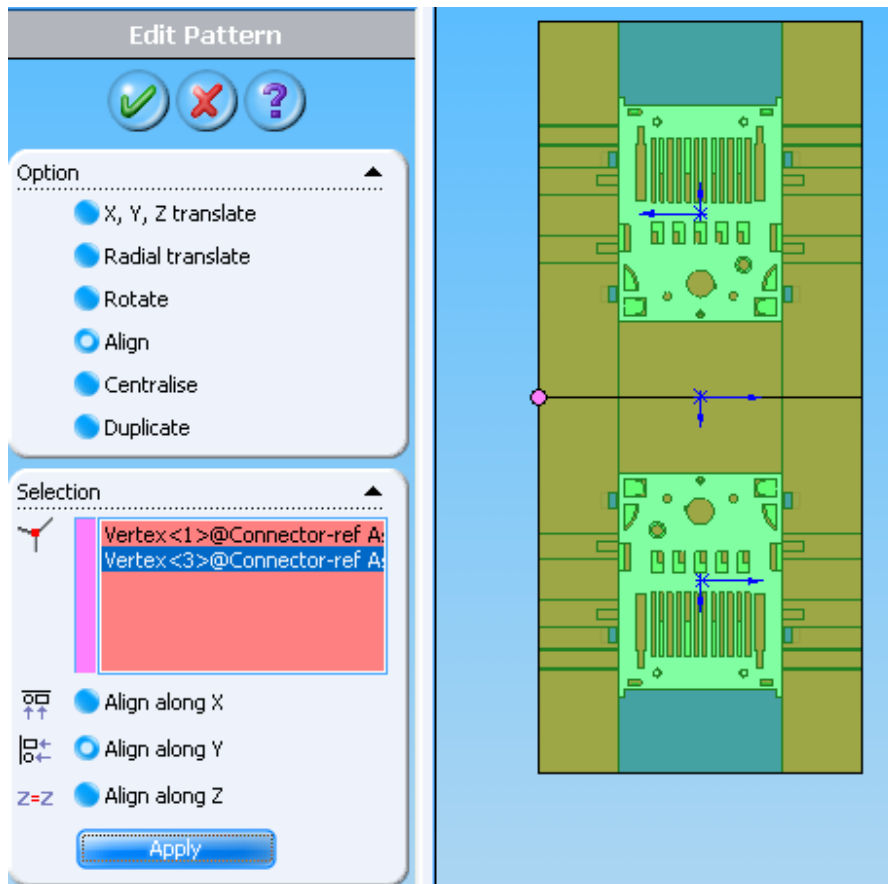
Align is in sequence , the first selected point or few vertex (moves) aligns to the last selected vertex (stationary) ;



For example, selected the vertex of the cavity to be translated and the destination vertices in sequence.



Click Apply, the cavity position will update.

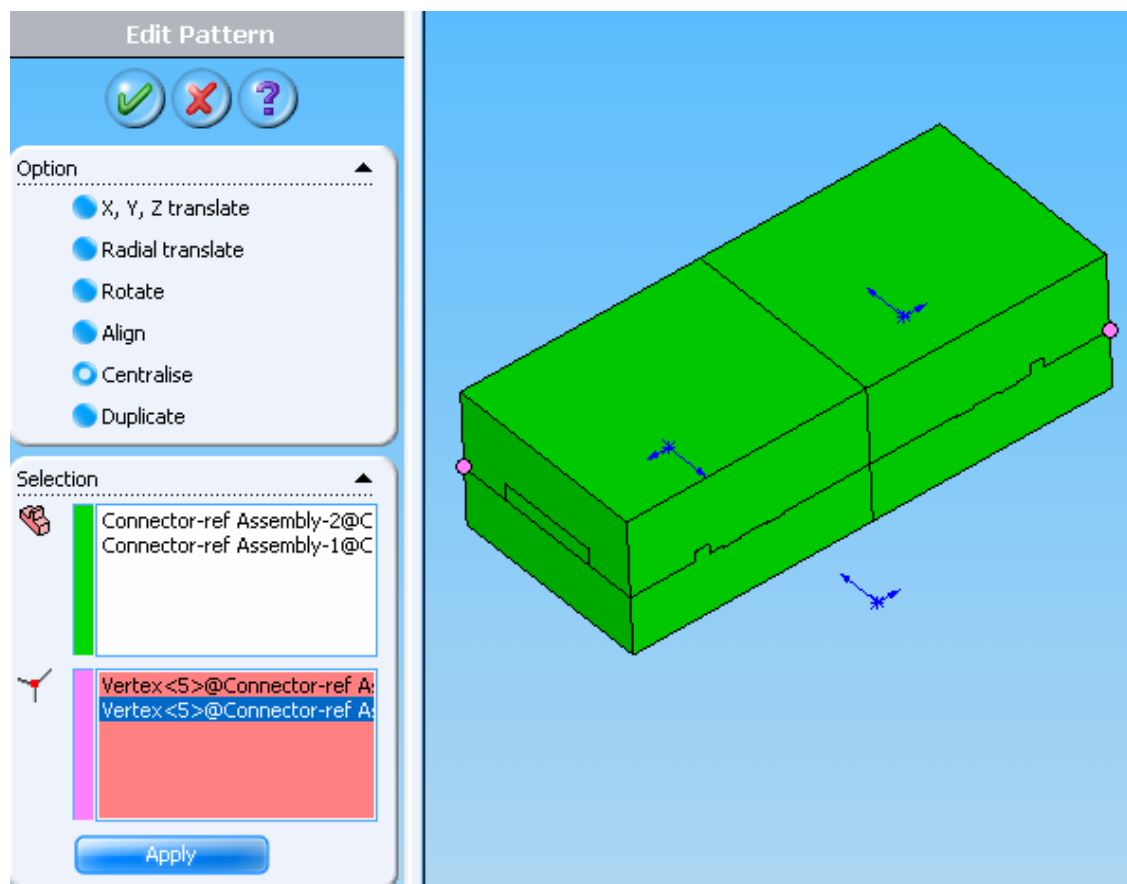


Centralize: Adjust the centre of the layout

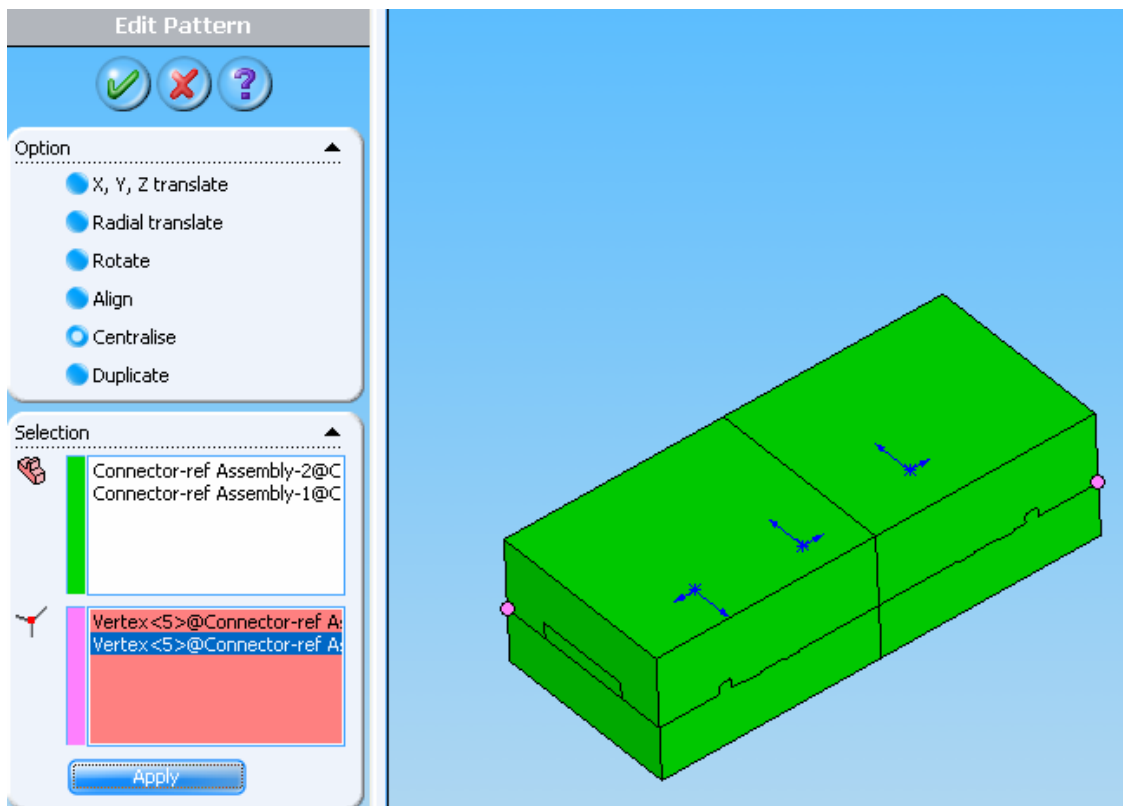
The center point of the polygon formed by the selected points is aligned to the Origin.

This is an important step as the center point has to be adjusted after the cavity is being edited.

Select cavity and reference point as follows



Click Apply



Duplicate: Duplicate the selected cavity.

Different situations will appear if Mirror copy is checked or unchecked.

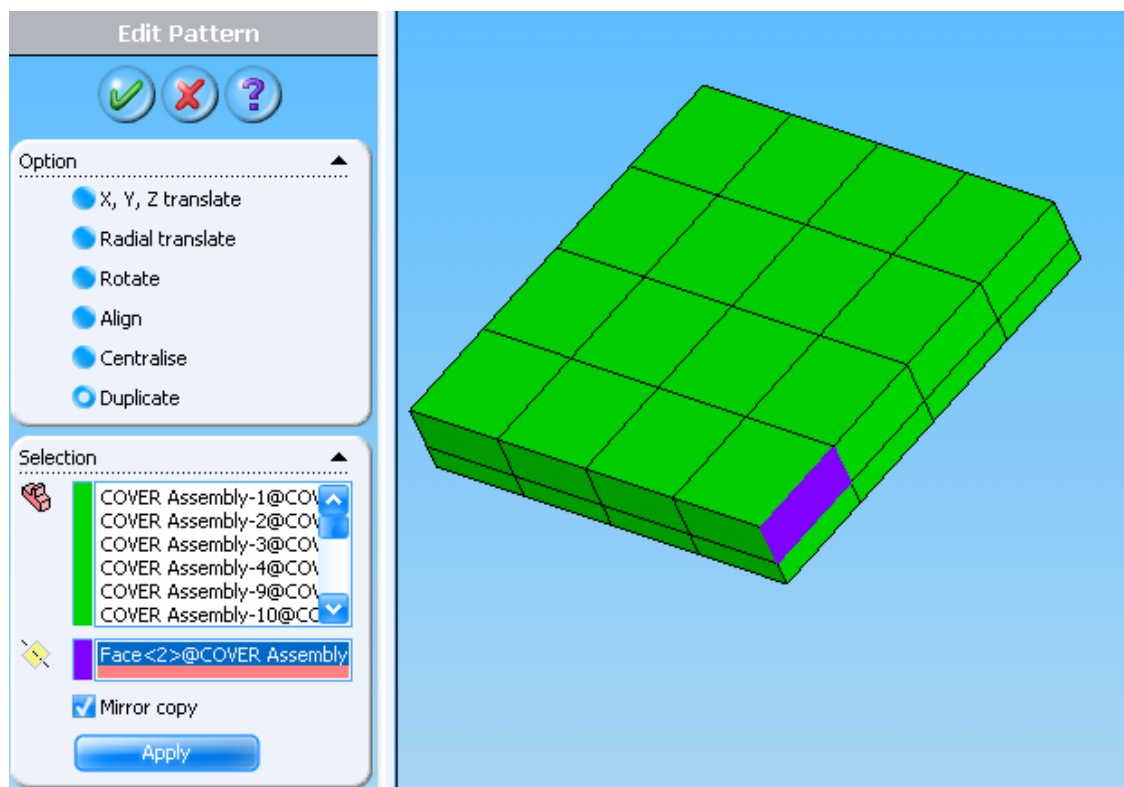
- Mirror copy is unchecked

Select the assembly to be duplicated, click Apply to proceed. The duplicate is overlapped with the original assembly.

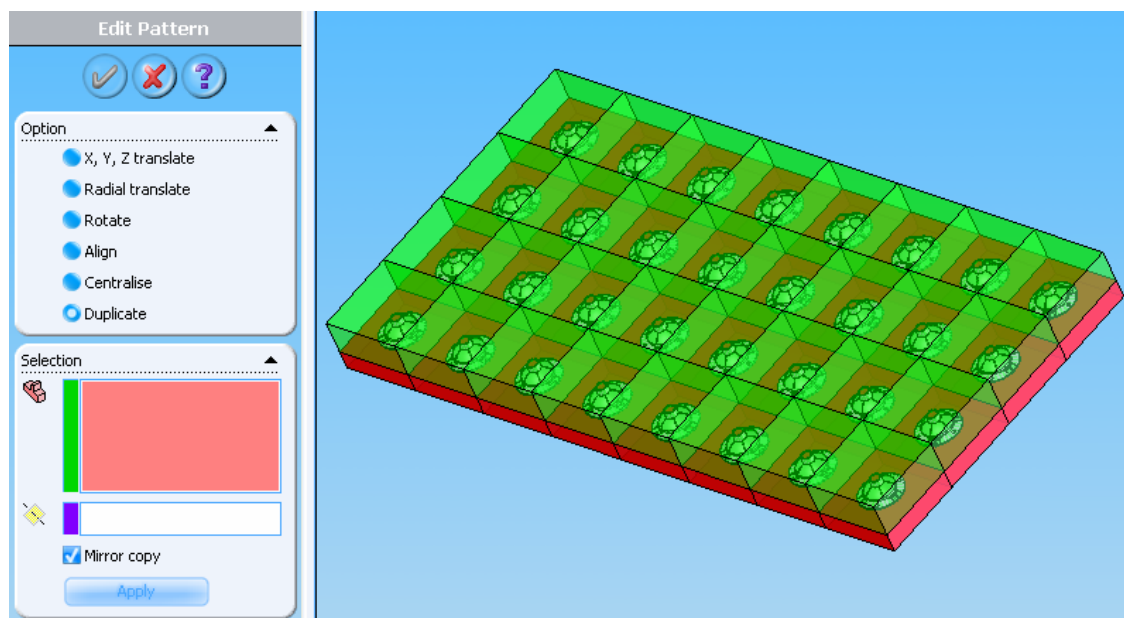
- Mirror Copy is checked

Perform mirror copy to the selected cavity, any planar surface can also be selected as mirror face, this can quickly produce 32-cavities, 64-cavities mold, etc..

As shown below




32 cavities layout as below

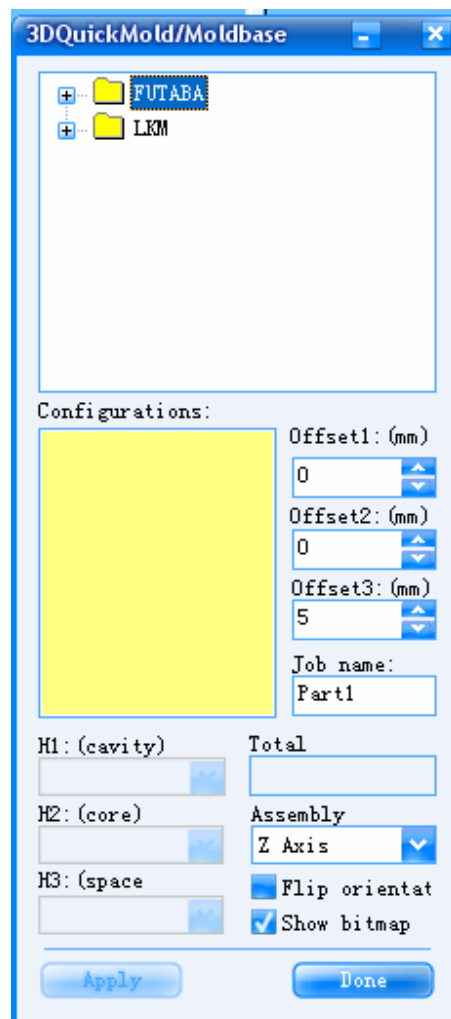


6. Moldbase Manager

The moldbase manager can:

1. Insert moldbase into the top assembly after Layout
2. Edit moldbase after the moldbase was inserted

After Layout, * Project.sldasm(top assembly file) is generated, Click , the Moldbase dialogue box pops out.



Standard moldbase is available in this dialogue box, the data listed is provided by the suppliers based on the published catalog.

The complete moldbase can be designed using the Moldbase manager, including standard and customized moldbase. The center of mold layout coincides with the moldbase center.

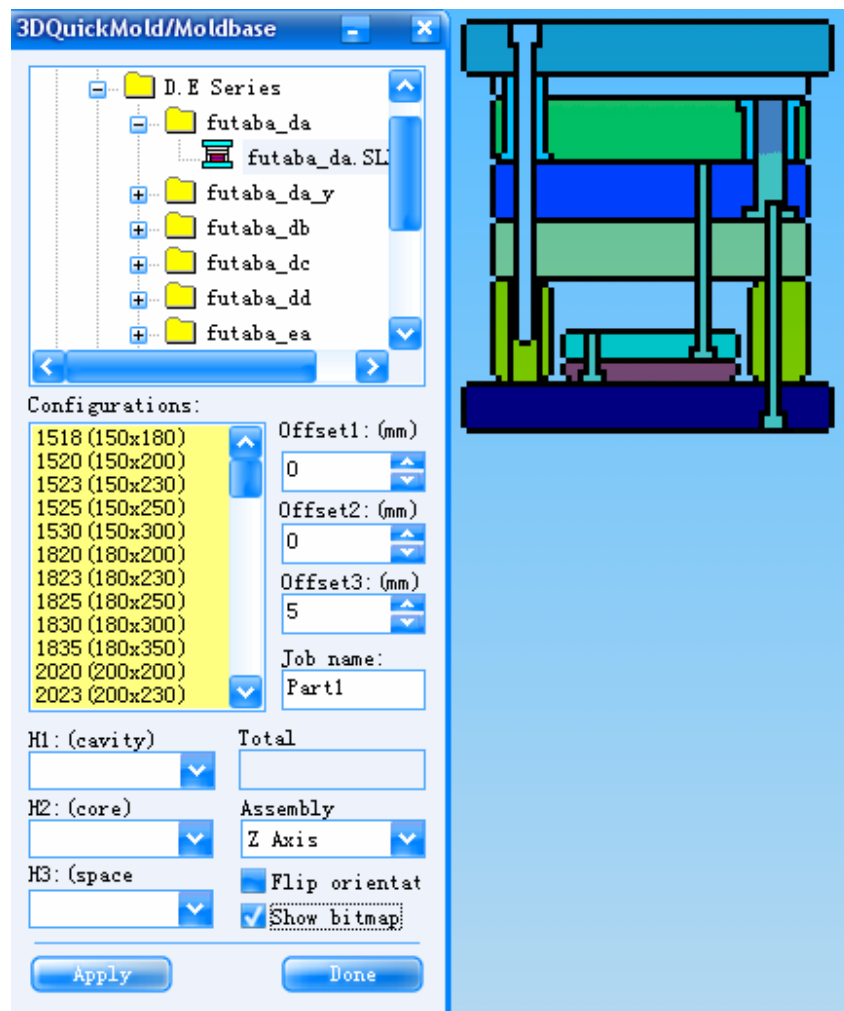
Typical design process as follows

Moldbase supplier—» Mold base type—» General dimension (L & W)—» H1, H2 and H3—» Gaps between fixed and movable half

In 3DQuickMold, standard Moldbase from FUTABA and LKM is available for selection, more standards will be provided in future. Customized moldbase could be added.

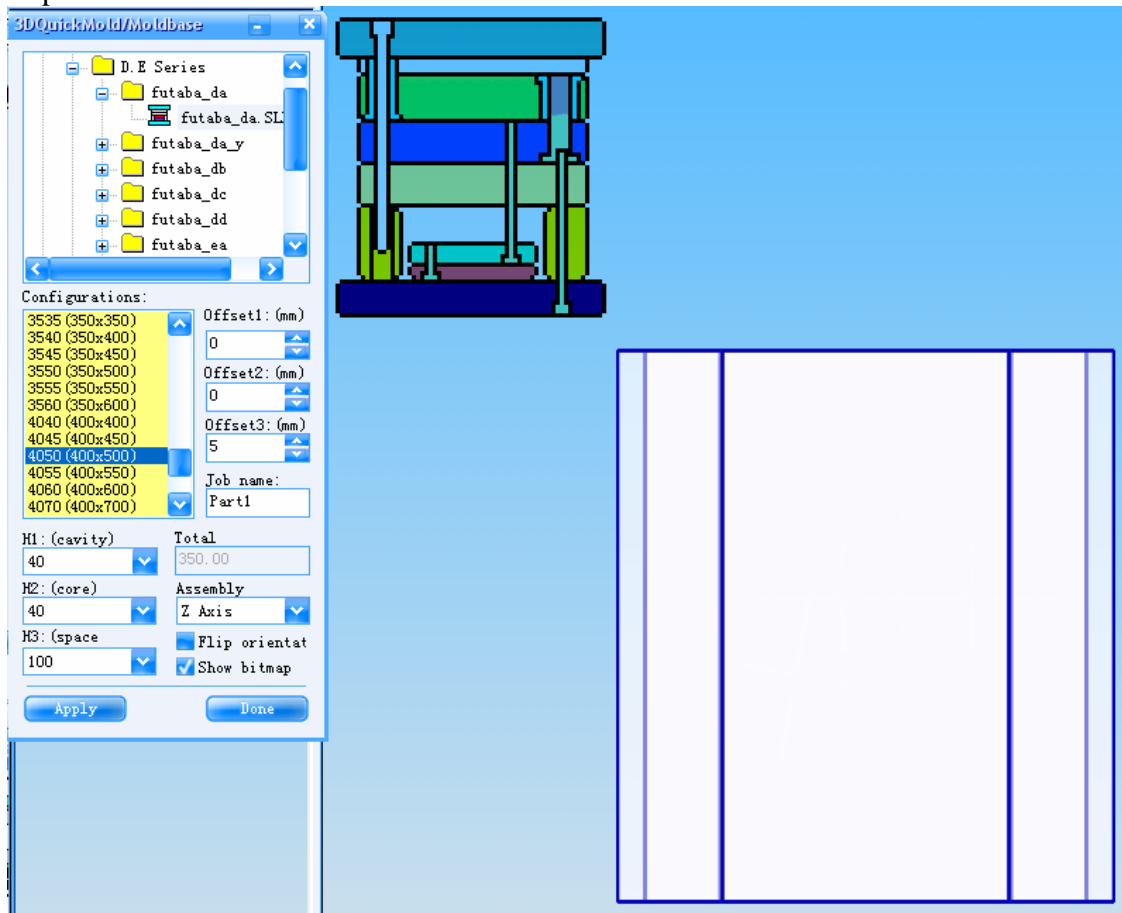
To construct the moldbase, first select a standard moldbase, then edit the parameter of the moldbase.

Select a type of moldbase, the preview of the selected moldbase appears at the top left corner of the graphic area (check Show bitmap at the bottom right corner of the dialogue box to preview). As shown below.

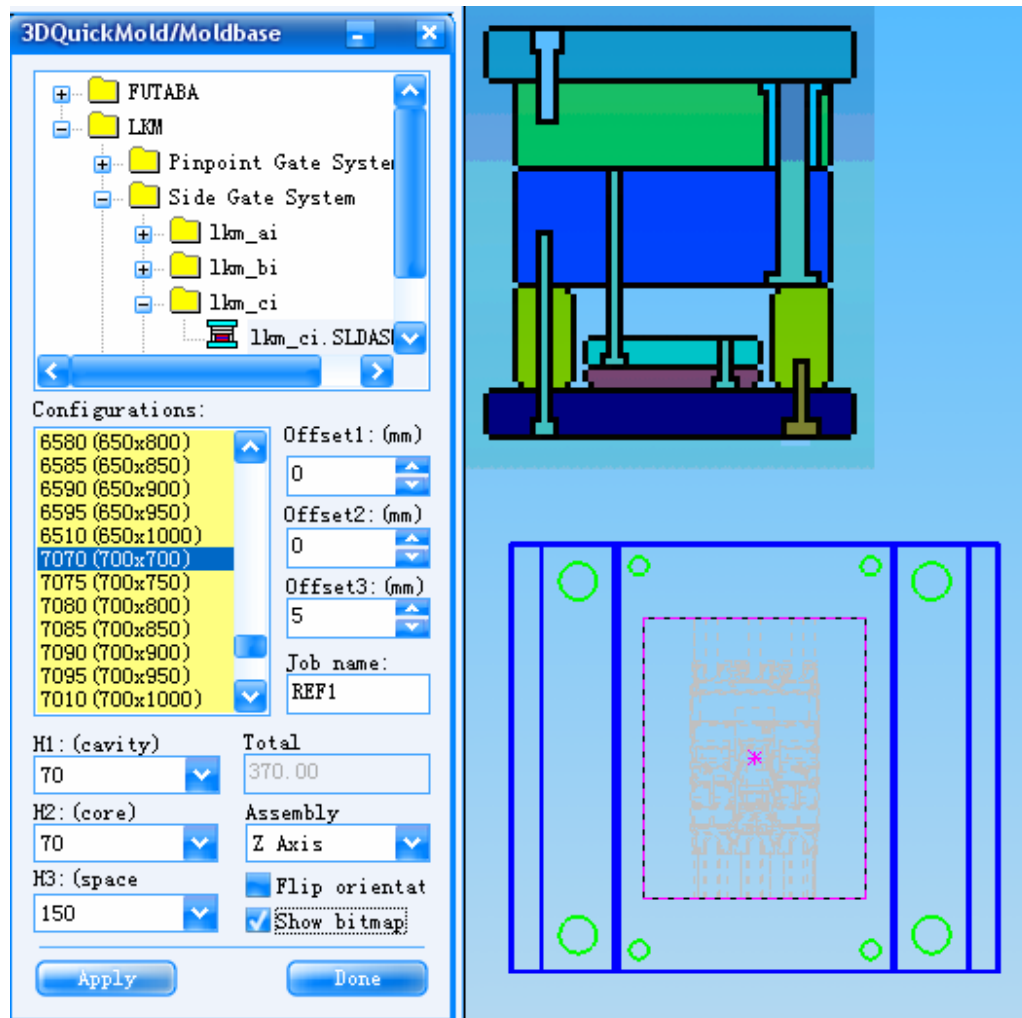


Under Configurations, different dimensions of the moldbase are shown. Select one of them, a 3D preview appears at the graphic area. This can be view in different orientation.

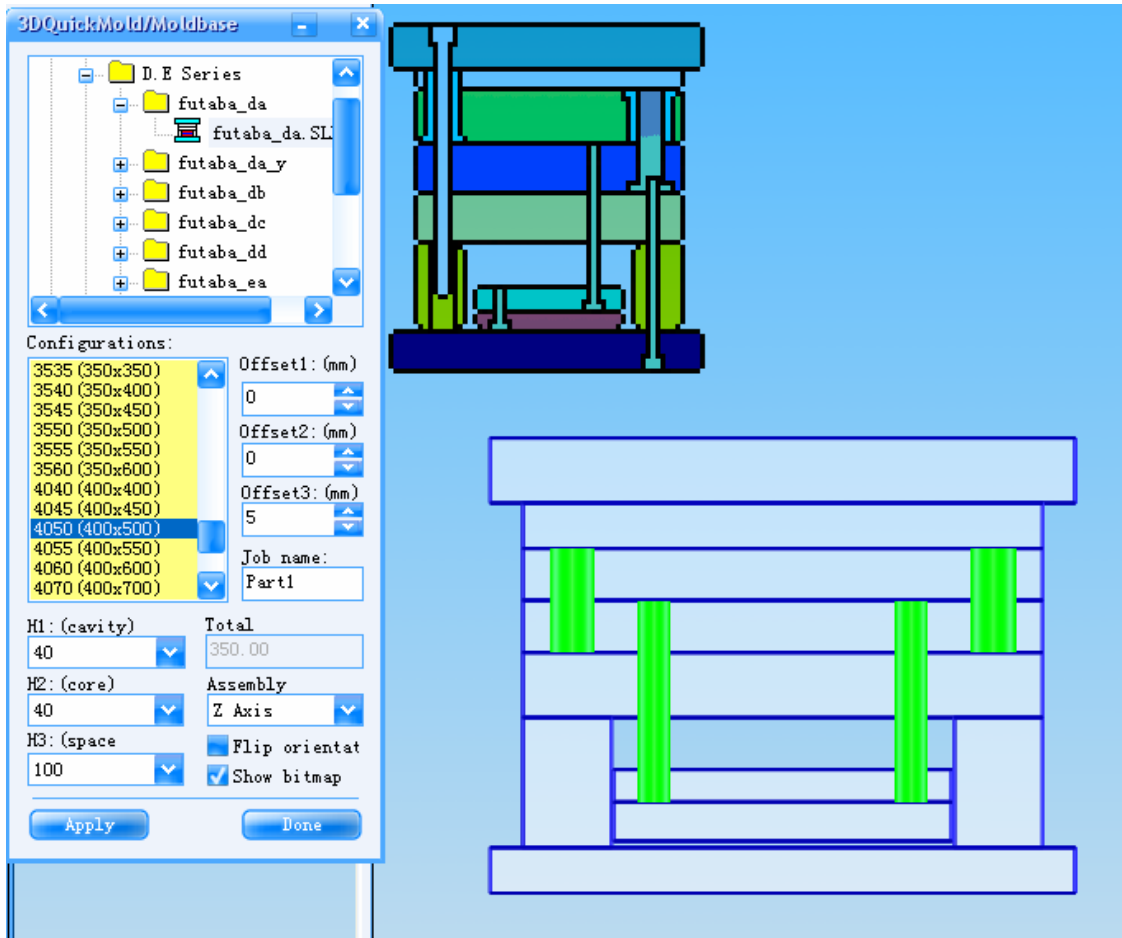
Top view



Use the Wireframe option to show a clearer view.



Front view



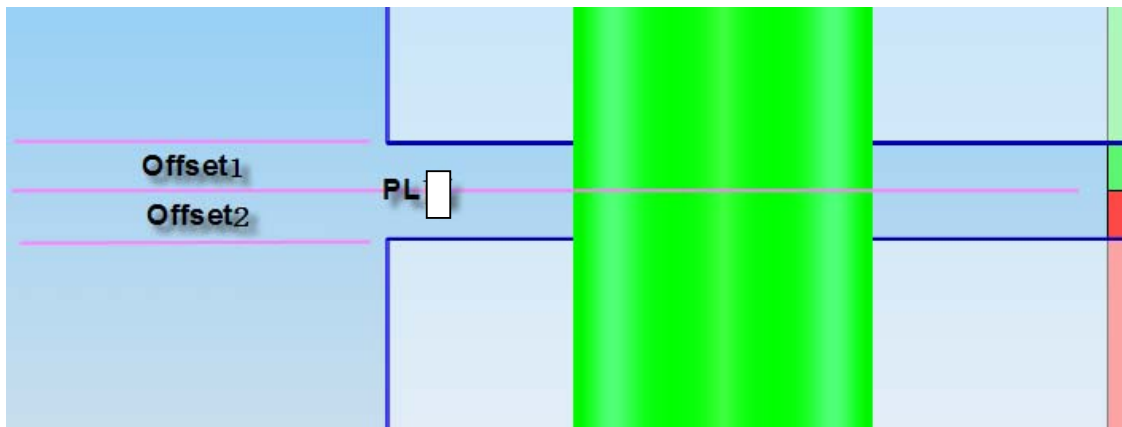
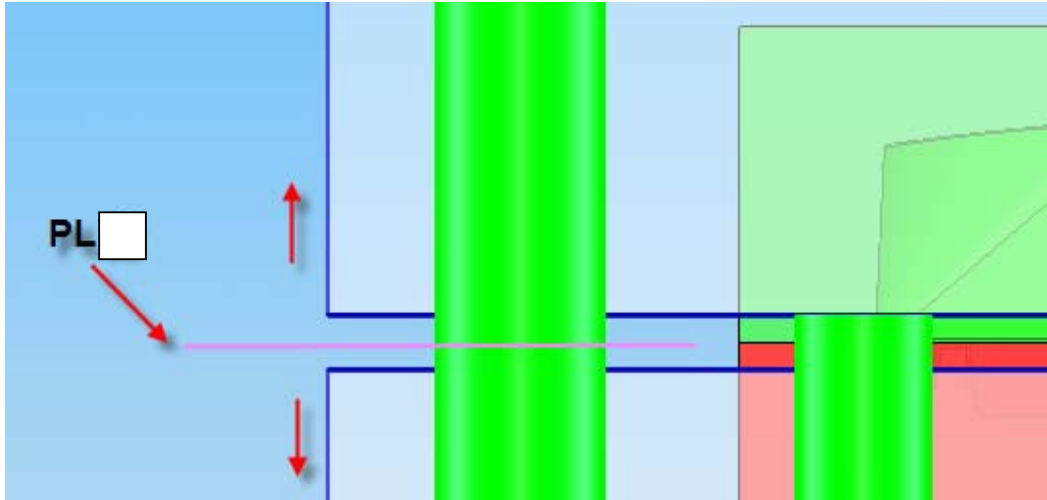
H1 (cavity): Define the thickness of A plate.
Standard thickness is available in the pull down menu.
All the value is standardized, custom input is not allowed at this time.

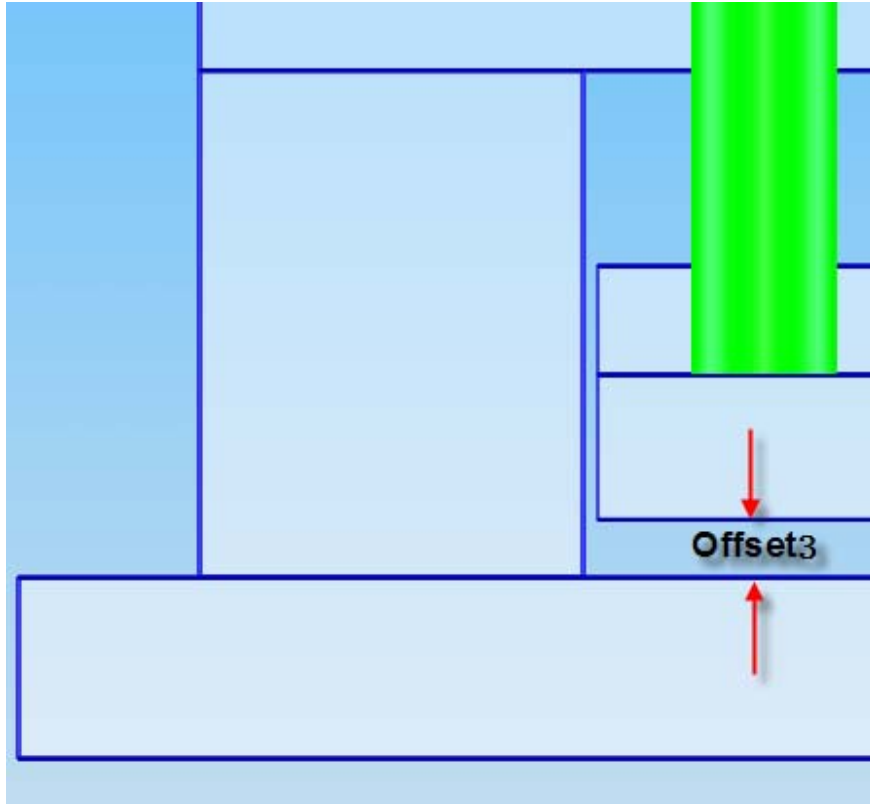
If custom input is required, first arbitrarily select a value to generate the moldbase, then click Moldbase to enter the edit mode to edit.

H2(core): for B plate
H3(space block) : for C plate

To edit the thickness of other plates, first generate the moldbase, and then click Moldbase again to perform editing.

Offset1: The spacing from cavity plate to moldbase center
Offset2: The spacing from core plate to moldbase center
Offset3: The spacing between ejector plate and bottom plate





Job name: Prefix for all components in the mold base
 Assembly will not be affected.

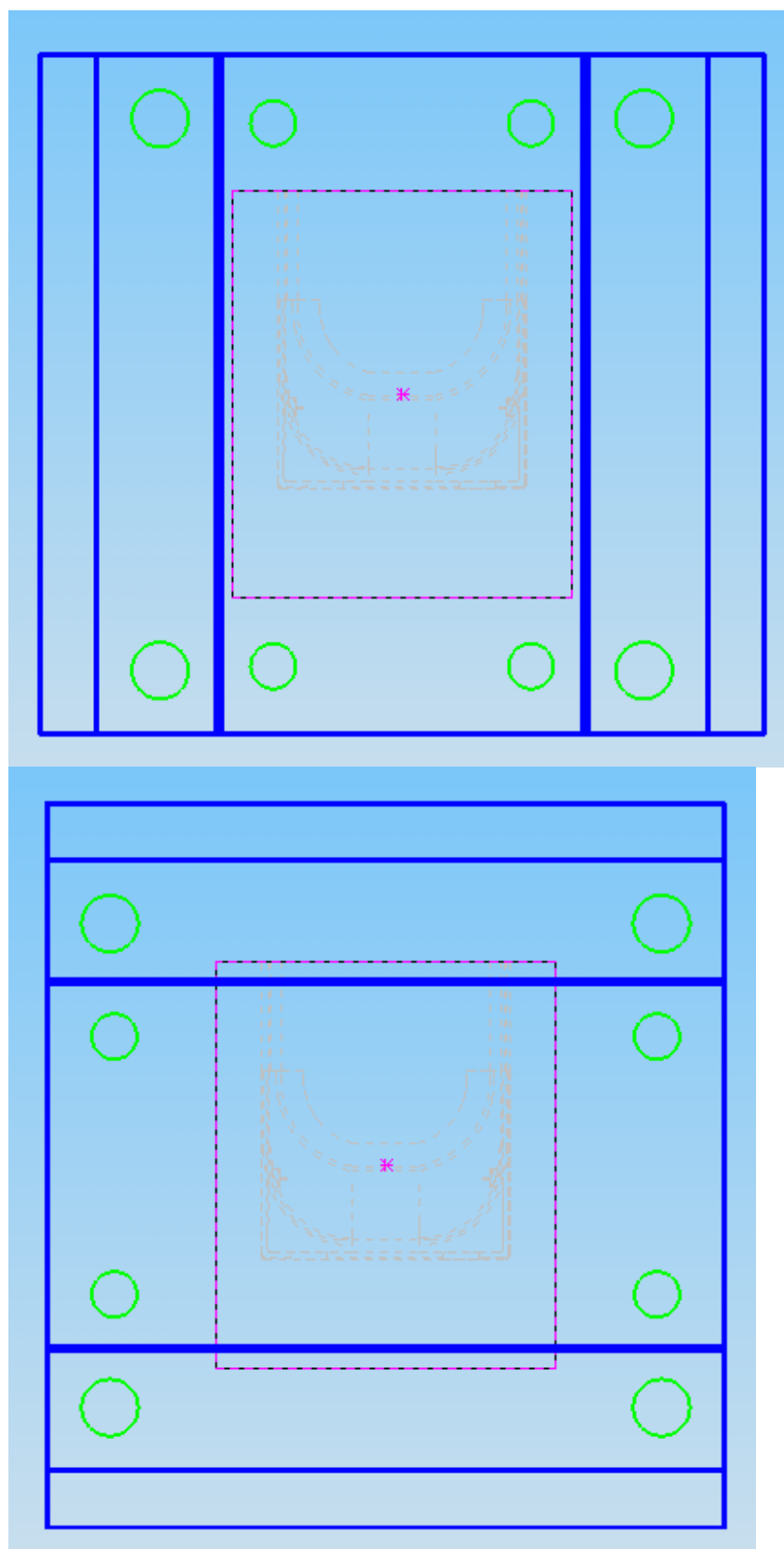
Total: Height of the mold base, this value for reference only, it cannot be edited directly.

Assembly: The relation between the mold layout and the mold base
 There are altogether 6 directions, the X、Y、Z、-X、-Y、-Z direction for selection
 The preview can be seen in the working window
 This option is used for the part not using +Z direction for assembly.

Flip orientation: Adjust the relative direction of the moldbase and mold layout.

The following pictures show the two different situations.

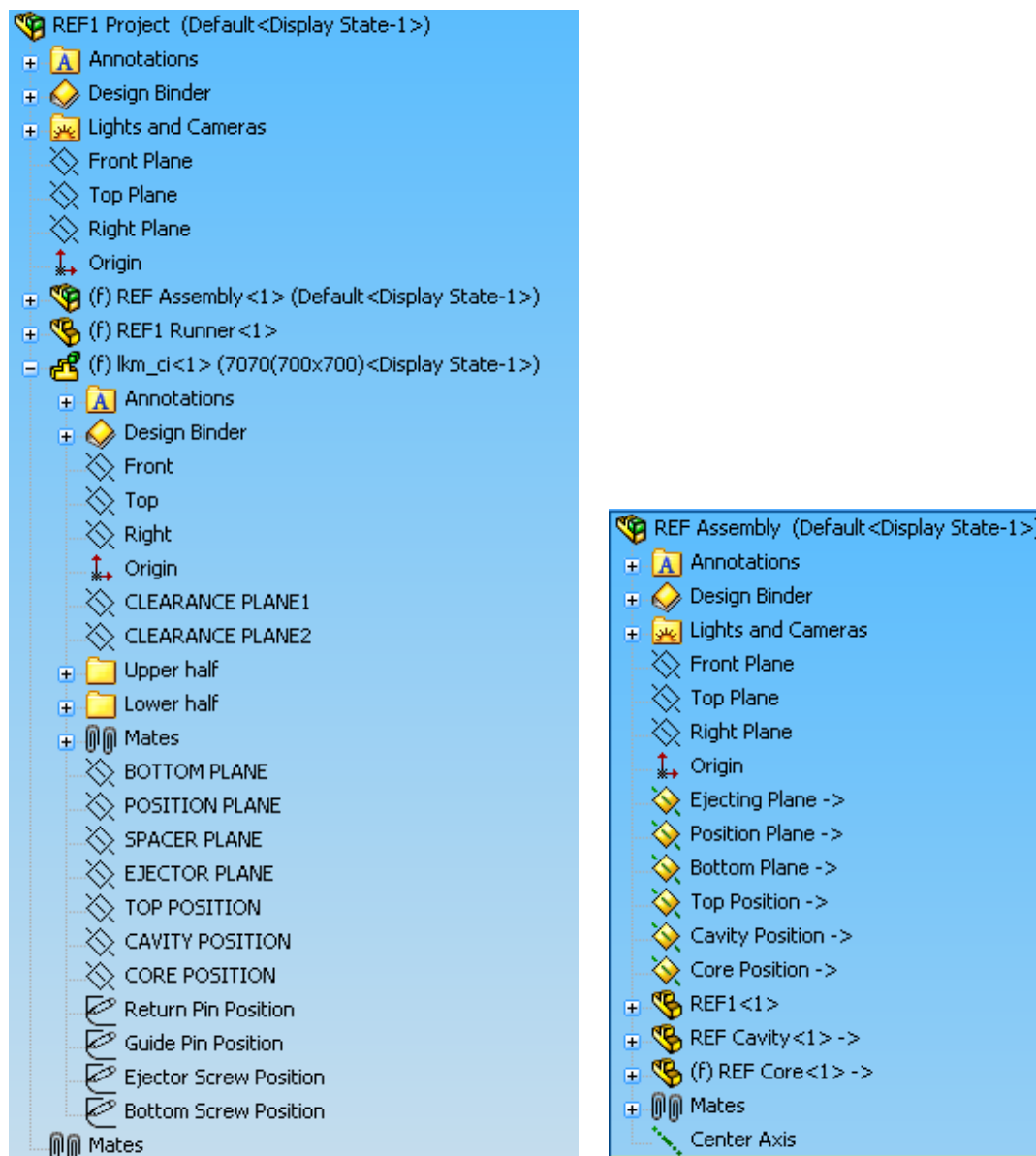
Flip Orientation is checked or unchecked.



Show bitmap: Show bitmap at the top left corner of the working window.

After setting up, click Apply, 3DQuickMold will generate the moldbase
The moldbase generated will be stored in the current working directory but not the default directory of the 3DQuickMold. It may take time to finish as there are many parts. Click Done to quit from the dialog.

A moldbase assembly appears under the * Project.sldasm tree. All reference plane related to the moldbase will be copied into * Assembly .sldasm. Those planes will be used for place some components such as ejectors and mounting screws.

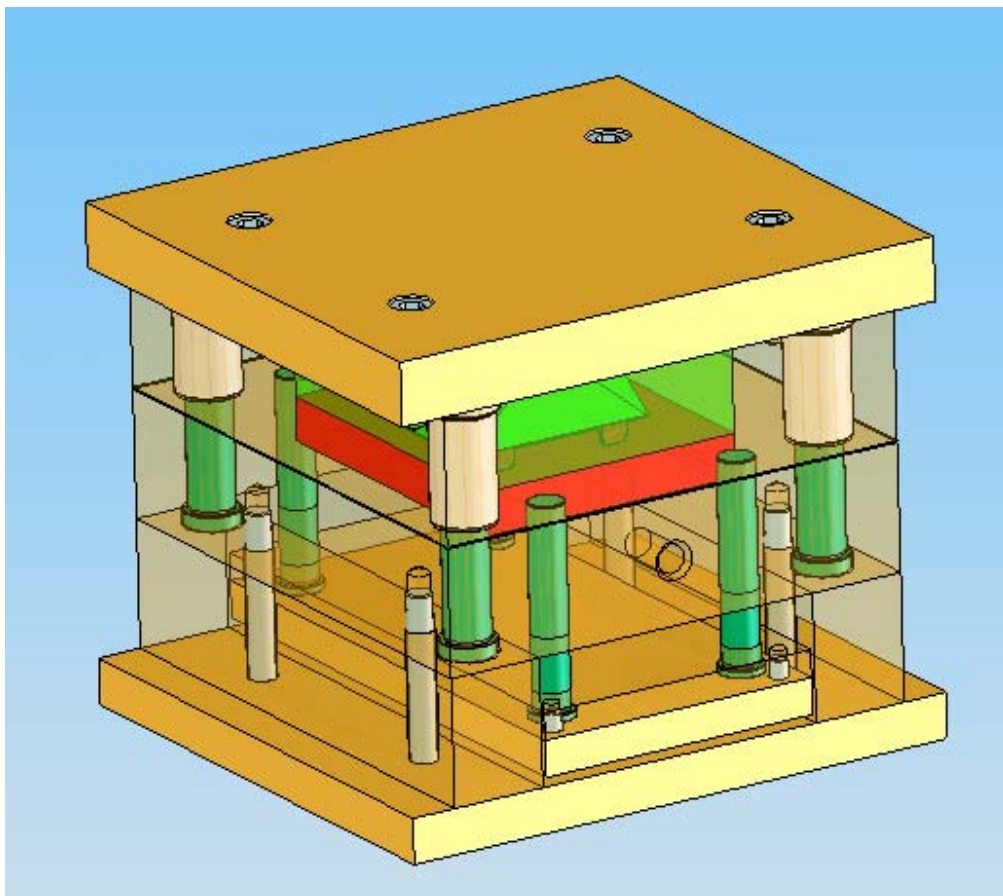


The moldbase assembly includes:

1. reference planes to represent some important moldbase positions
2. all moldbase components such as mold plates and screws
3. sketches to control screws, guide pins, return pins and etc.

Feature of Moldbase

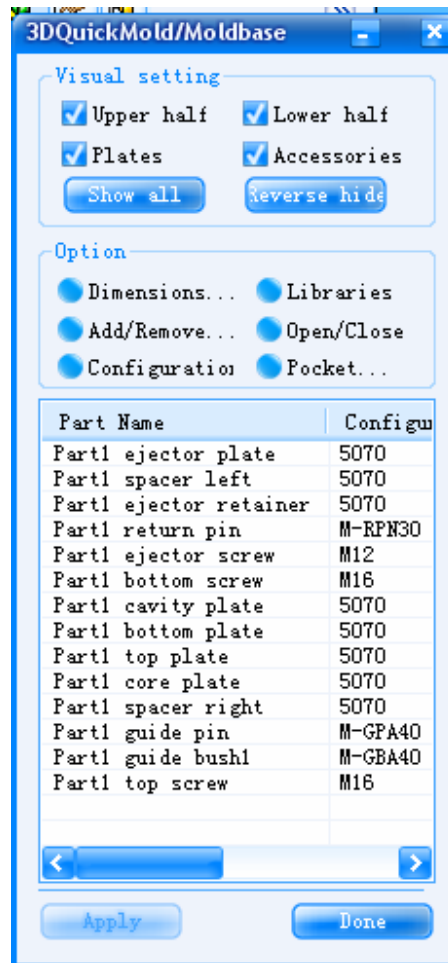
- native Solidworks features
- mate constraints based on mold open/close movement
- support direct editing
- pockets ready



After the mold base is added, click the ***Moldbase Manager*** to enter the Moldbase editing function, current working window will automatically switch to the Moldbase assembly.

To construct non-standard moldbase, first select a similar standard one, and then do the custom editing. The following situation can be carried out in the edit mode.

- changing the dimension of the mold plate and the position of the standard parts.
- add or remove extra mold plates. (such as manifold plate, double ejecting mold)
- insert additional standard parts.



Visual setting: set the visibility of different mold base component;

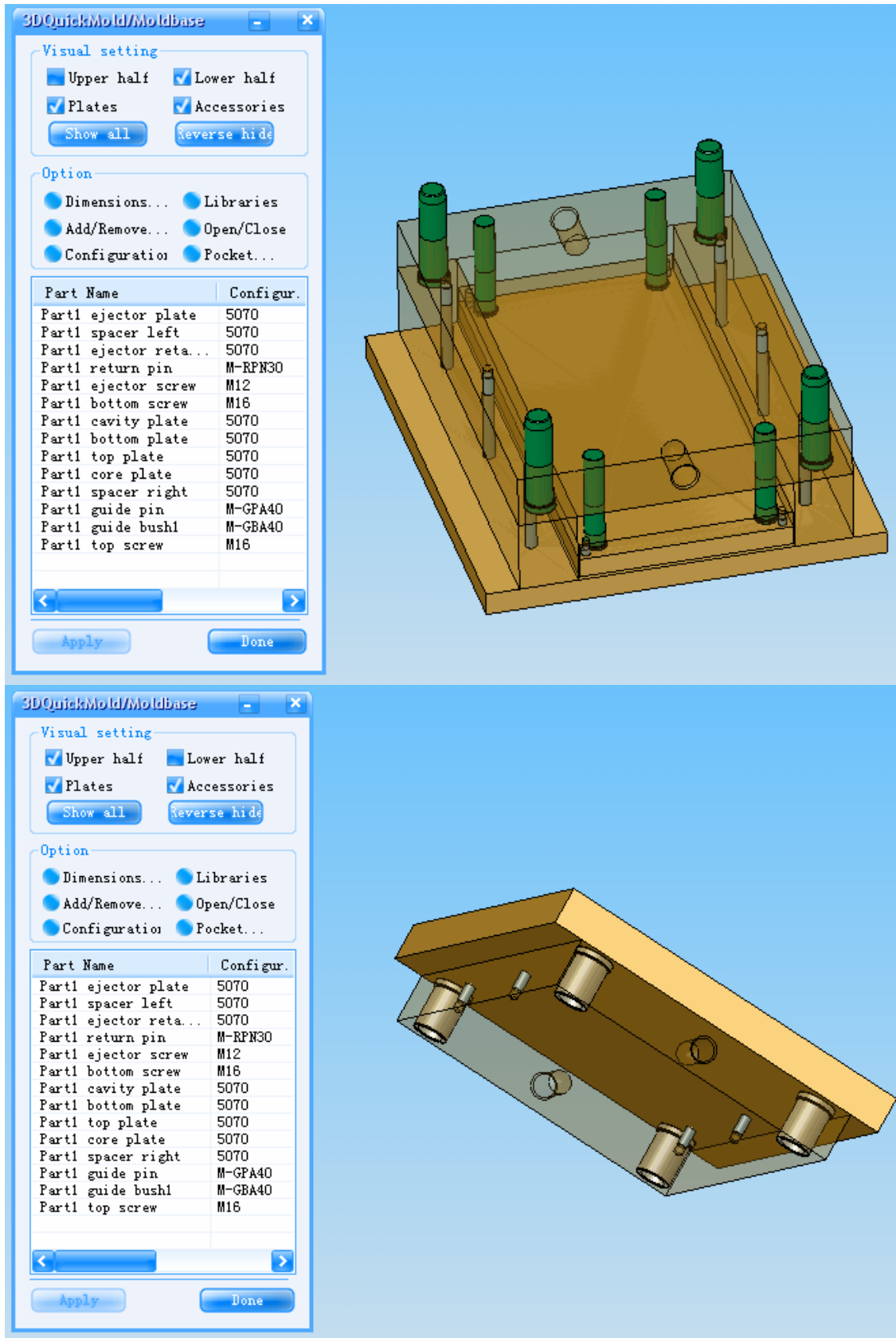
Upper half: show/hide the Upper half

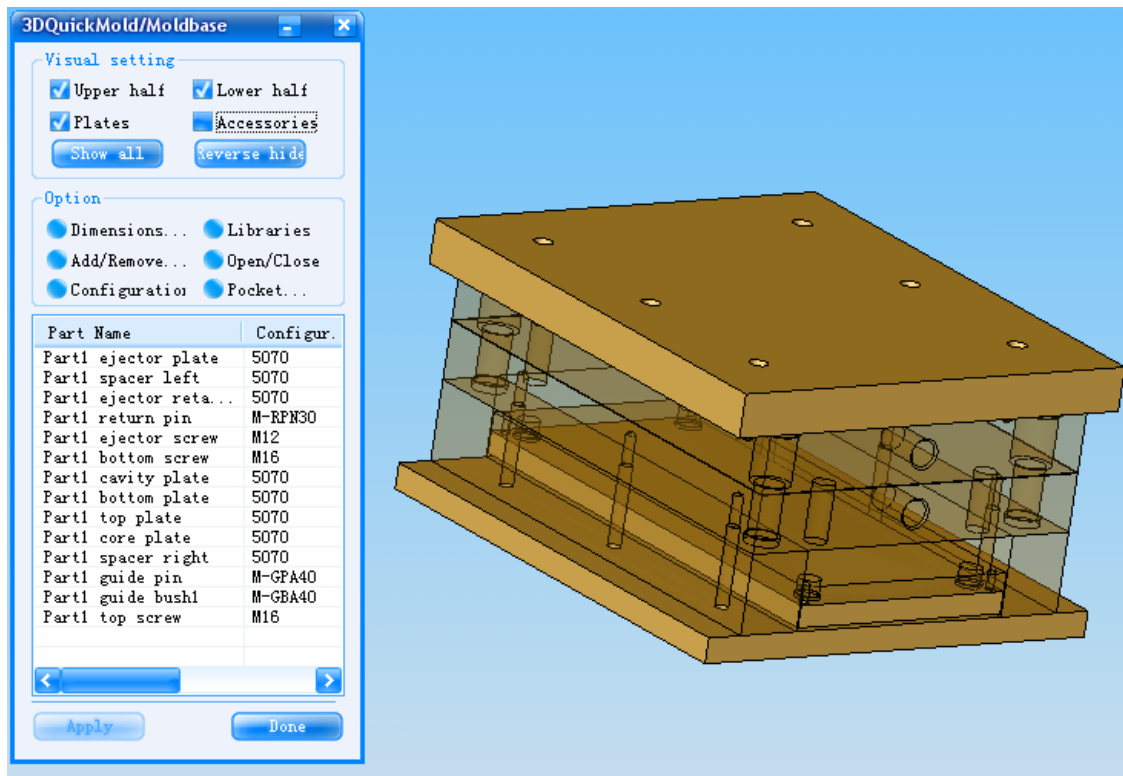
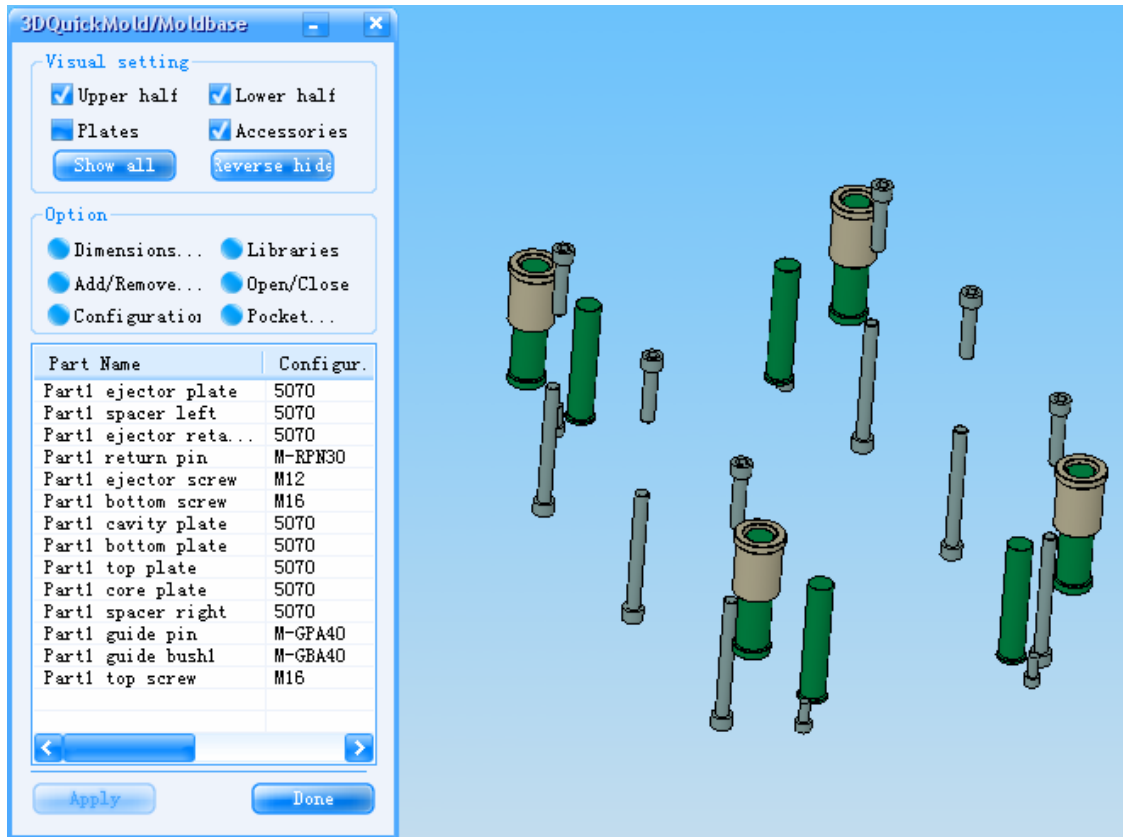
Lower half: Show/hide the Lower half

Plates: Show/hide the mold plates

Accessories: Show/hide the accessories (guide pins、guide bushings、screws)

The details are as follows:





Show all: Show all components.

Reverse hide: Show only the selected parts. Select a few components and reverse hide the unselected.

Hiding the other parts facilitates some operations like Pocket, Position or add Accessories to the moldbase assembly.

Under Option, there are 6 options:

Dimensions

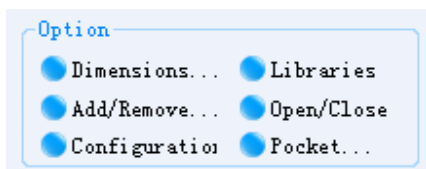
Libraries

Add/Remove

Open/Close

Configuration

Pocket



Dimensions

Dimensions





General size

 160.00mm

 160.00mm

☒ Flush moldbase

 160.00mm

 33.00mm

 90.00mm

 50.00mm

 40.00mm

 60.00mm

 2.00mm

 1.00mm

Component position

SP <==> GP

Return Pin Position
 Guide Pin Position
 Ejector Screw Position
 Bottom Screw Position



 200.00mm

Apply

Screw offset


Dimensions






General size

 160.00mm

 160.00mm

☒ Flush moldbase

 160.00mm

 33.00mm

 90.00mm

 50.00mm

 40.00mm

 60.00mm

 2.00mm

 1.00mm

Component position

Screw offset



Δx 0.00mm

Δy 0.00mm

Apply

Reset

Dimension of all the plates of the moldbase can be edited;









: length of moldbase



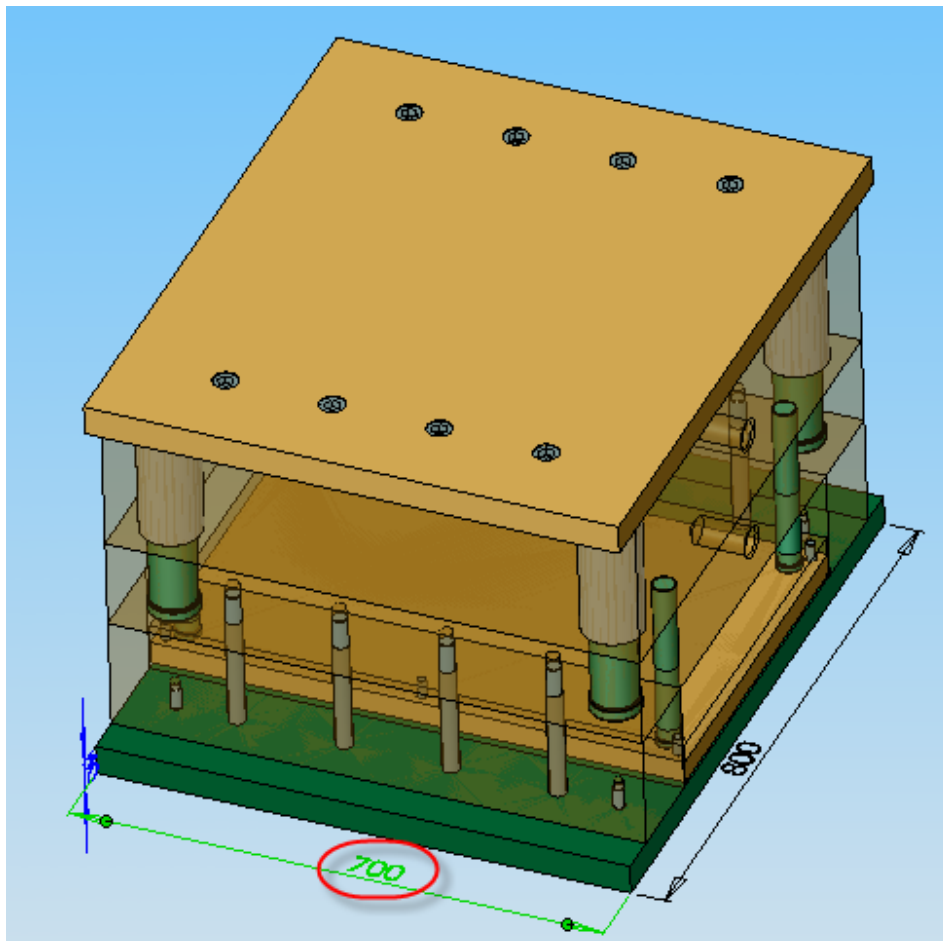
: width of moldbase;

Flush moldbase: Check this option, An I type moldbase will become H type.

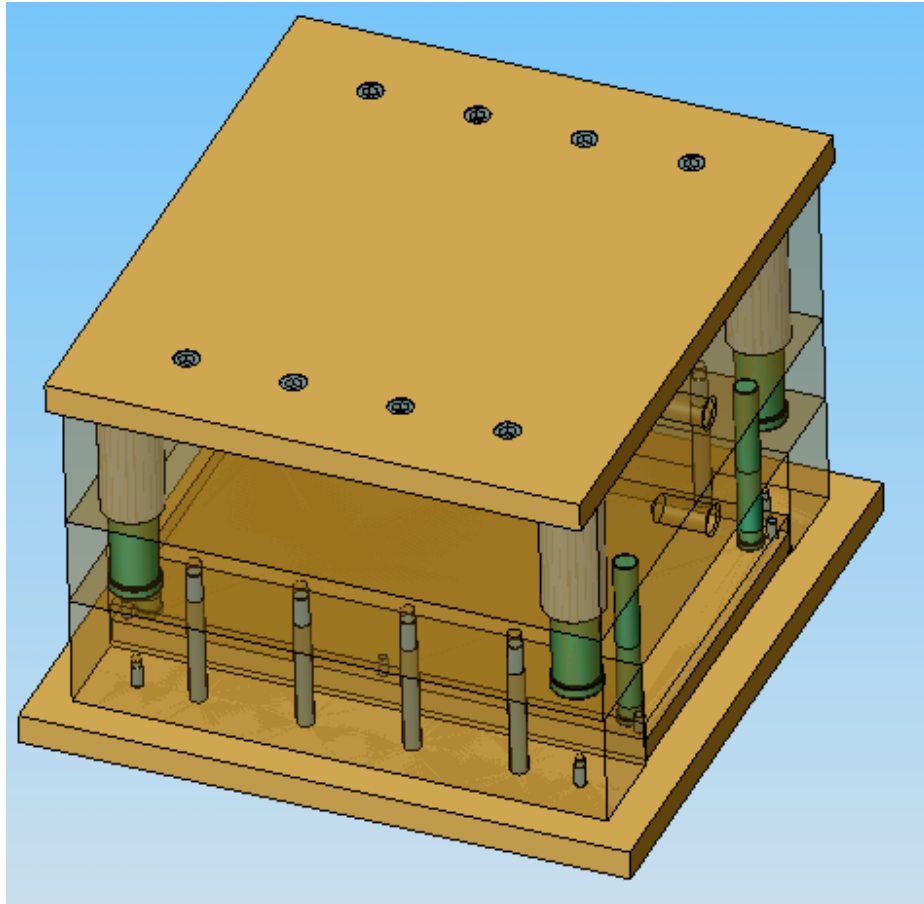
-  : width of bottom plate;
-  : thickness of spacer block;
-  : width of ejector plate;
-  : thickness of plate A;
-  : thickness of plate B;
-  : height of the spacer block;

Notes: For some dimensions not listed in the dialog, user can double click the component in the graphic area to display its standard dimensions and do the editing. After editing, clicks rebuild or go to the property tree to perform the edit dimension.

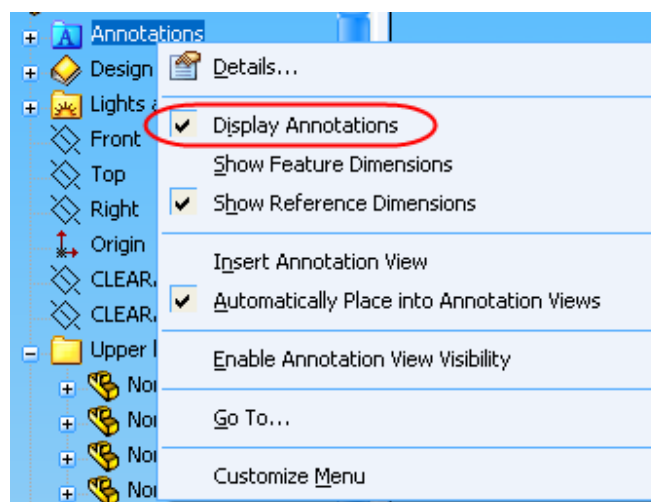
For example: double click the bottom plate to display all dimensions, change the dimension from 700 to 800.



Rebuild model



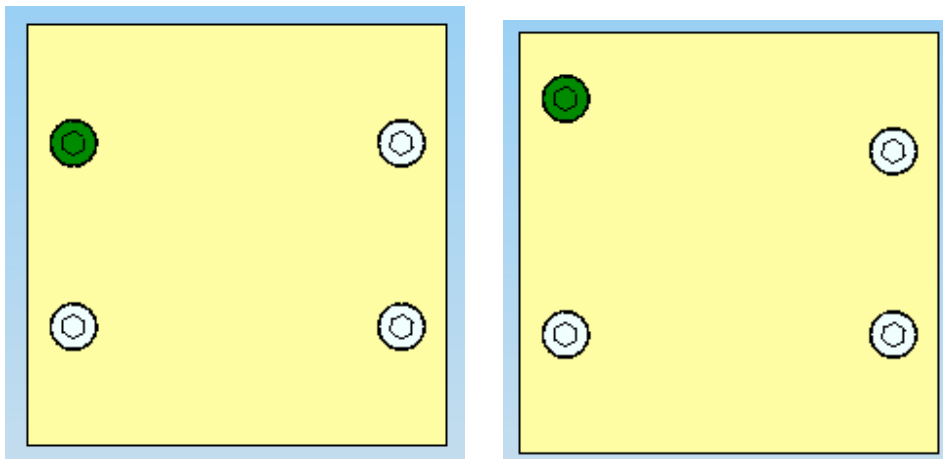
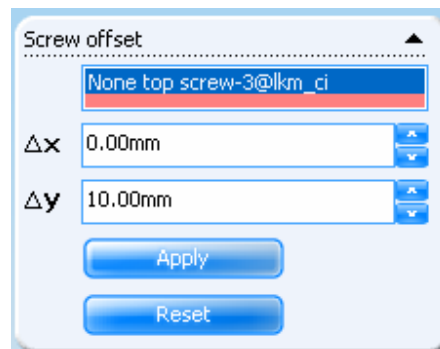
Component position: Edit the position and dimension of the accessories.
Make sure that Display Annotations option is checked.



Screw offset: Edit the screw position.

This function can only edit the screw position on the line up direction (Although there are both X and Y direction for translation, only the direction along the length of the space block could be edit)

Click **Apply** to translate the screw, click **Reset** button to move back to the original position;



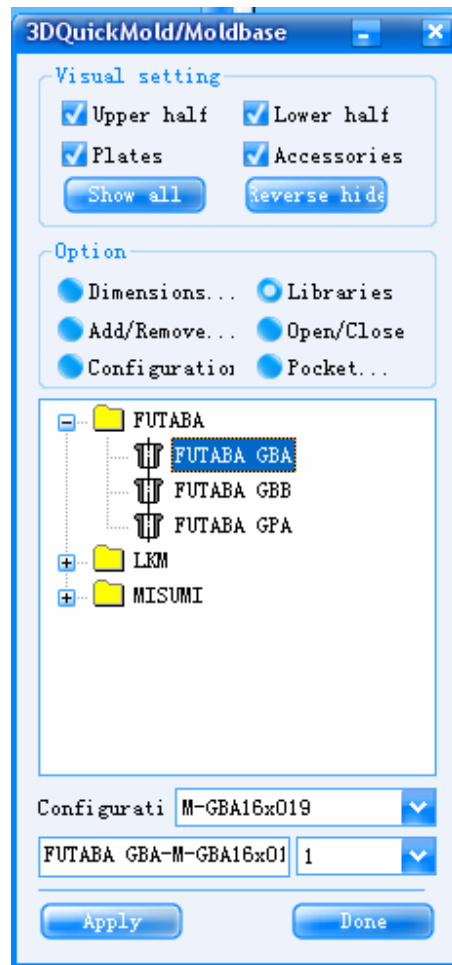
Libraries : Add standard libraries for moldbase

Select any databases, 3DQuickMold displays the entire relevant componets, drag and drop the parts onto the mold base

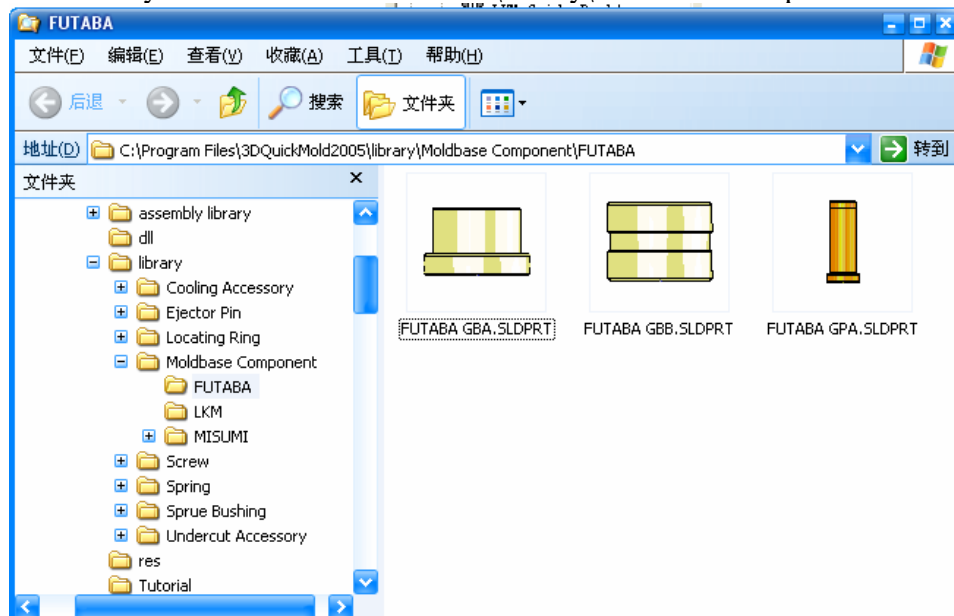
In **Configuration**, select as appropriate.

The next line is the part name,

Similar operation is available in the Library Manager, this option will be removed in future.





The Library is in the installation folder\library\Moldbase Component



Add/Remove: Add or remove selected mold plate (this function can produce multiple nonstandard mold base)

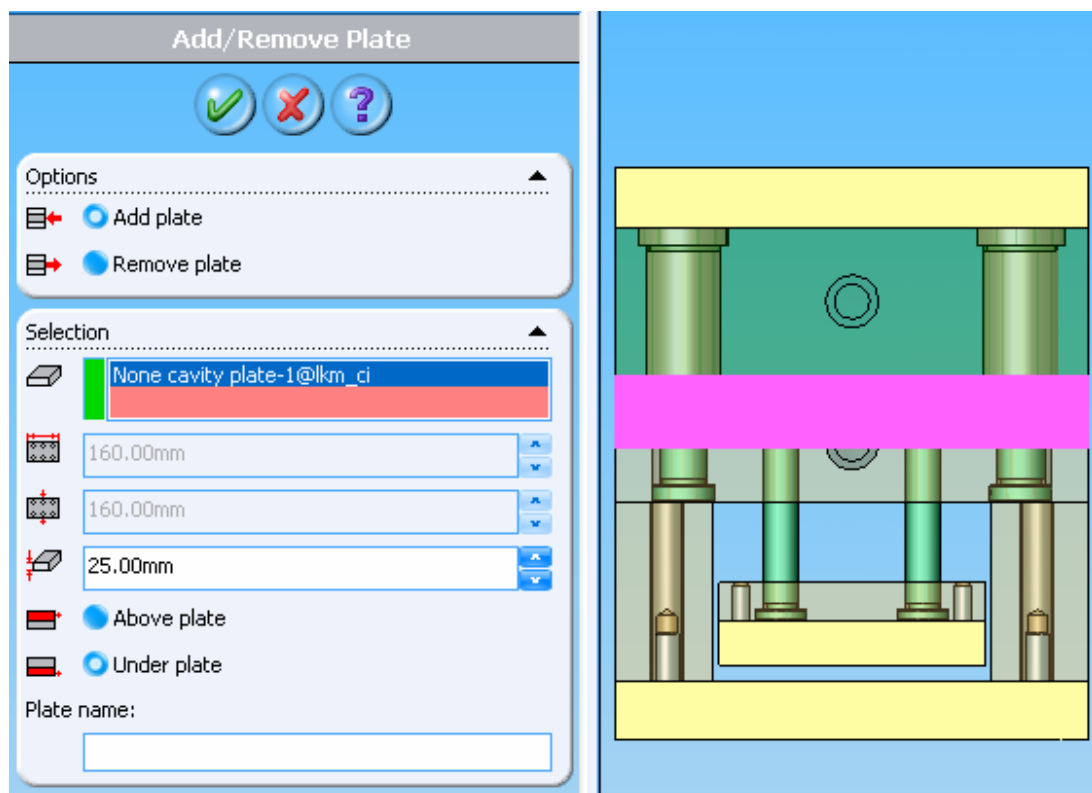
: add mold plate in the moldbase;

: remove the added mold plate in the moldbase;

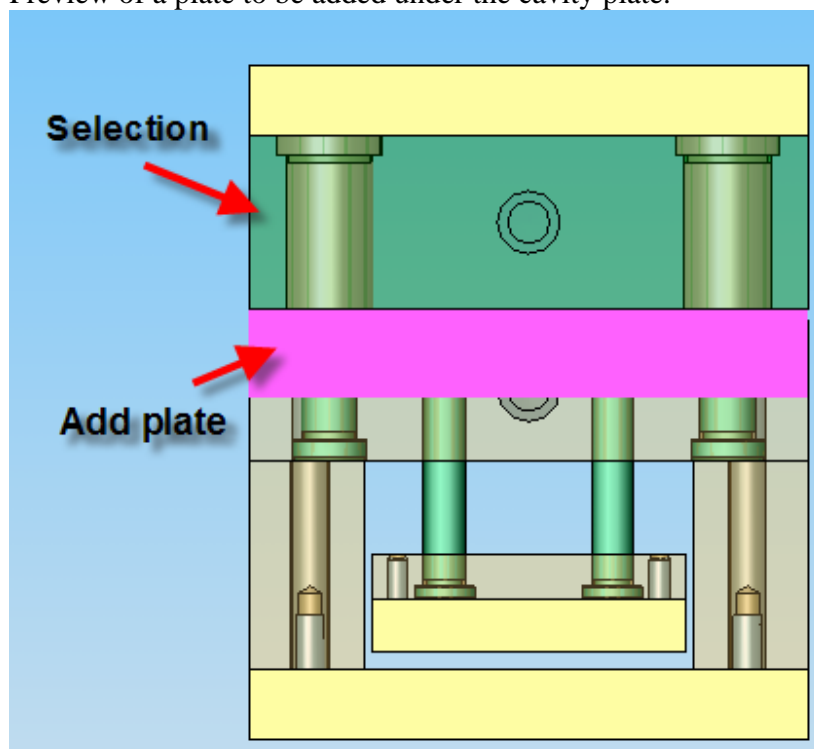
: reference plate, when adding mold plate, the mold plate will be added above or below the reference plate. Click to show the preview. If mold plate has to be removed, this would be the plate to be removed. the basic plate for the mold base could not be removed.

Note: when a mold plate is selected, 3DQuickMold decides whether a plate can be added above or below the reference plate. There are some rules for adding a plate:

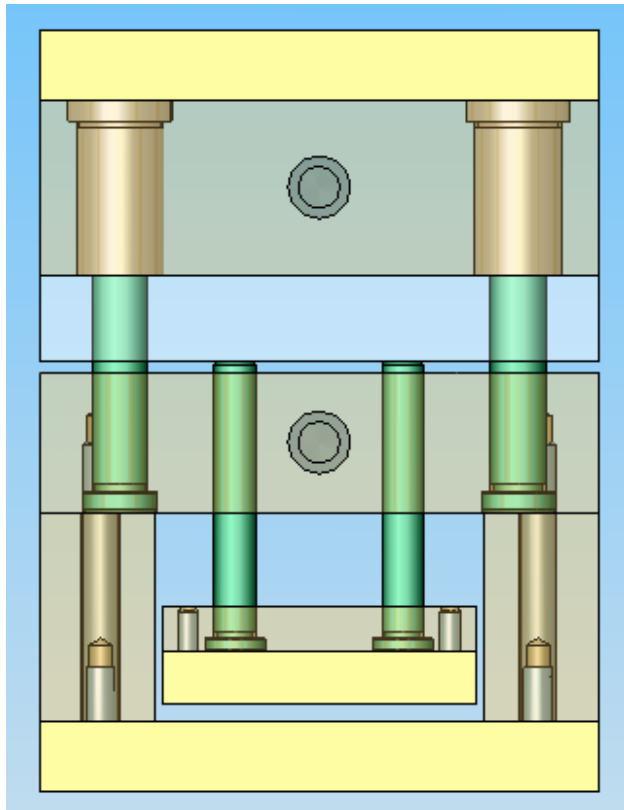
1. Top plate can only be added above it, so it the ejector plate.
2. Bottom plate can only be added under it.
3. Spacer block is not allowed to be reference plate.
4. Other plate can be added either above or below the reference plate




Preview of a plate to be added under the cavity plate.





Result after the plate was added.



: length of the mold plate to be added

As the moldbase is a standard moldbase, the length of the mold plate cannot be edited here, it is controlled by 3DQuickMold. The length here is for reference only.
The length here is defined by the reference plate.

: width of the mold plate to be added, it cannot be changed and for reference only;

: thickness of the mold plate to be added;

: add a plate above the reference plate

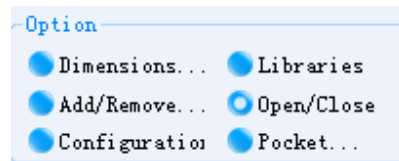
some types of plate cannot be added above the reference plate, this option is disabled for these types of plate;

: add a plate under the reference plate

some types of plate cannot be added under the reference plate, this option is disabled for these types of plate;

Plate name: Name the mold plate to be added;

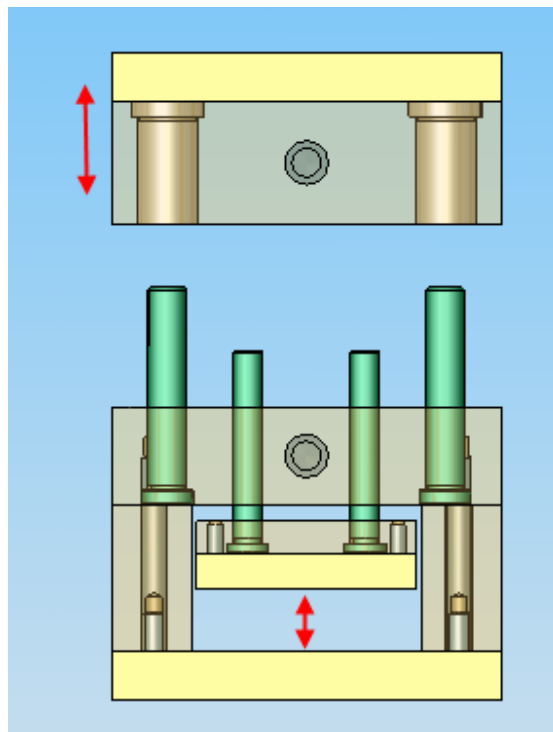
Open/Close: Show the mold Open/Close condition



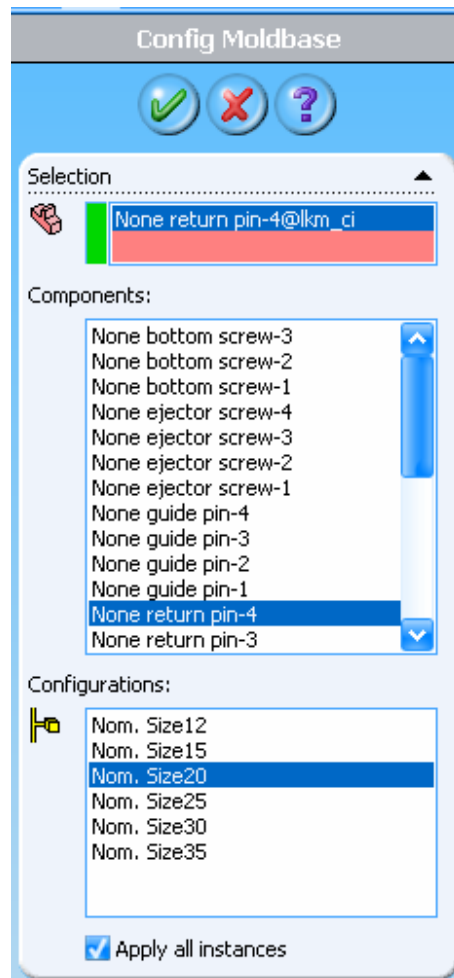
Open/Close: select ***Open/Close***, click ***Apply***.

The upper mold will move up, the upper mold and the ejector plate can be dragged, but only in the Z direction. The lower half is fixed and cannot be dragged.

Click Apply again, the moldbase returns to the previous condition, and it cannot be moved.



Configuration: change the configuration of a mold component. All the configuration of all the standard parts is listed. For example, if the mold base is enlarged, screw, pin, etc., have to be enlarged.



Selection: select the component to be edited;

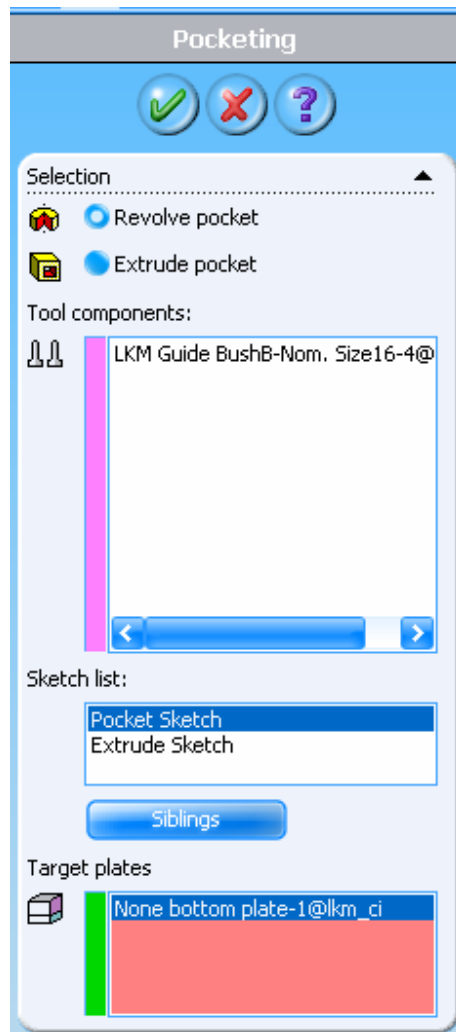
Components: select component form the list. The component is displayed in Selection

Configurations: relative configuration, if the configuration has to be changed, select and change the relative configuration

Apply all instances: Decide whether to apply the changes to the parts with same configuration.

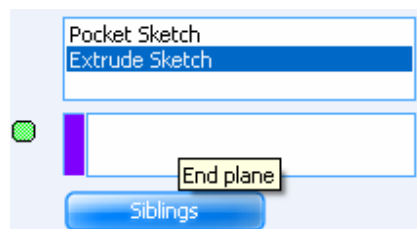
Note: after editing, some parts may not be changed. Click Rebuild to apply the changes.

Pocket: perform pocketing on the plates. Pocket can only be applied on the assembly;



Revolve pocket: perform revolve pocket on the plate. Revolved body like guide pin, guide bushes, screw and ejector pin can be pocketed using this function.

Extrude pocket: perform extrude pocket on the plate. The Extrude Sketch is necessary and the End plane should be selected as well.



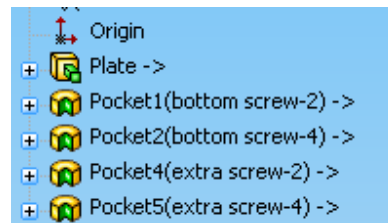
Tool components: tool components for pocking, for example guide pin, guide bush, screw, ejector pin, etc..., If many parts of the same type exists, select one of them, use ***Siblings to search and select all the parts of the same type.***;

Siblings: Select all the parts of same type at the same assembly level.

Sketch list: list of sketches to pocket


Target plates: target plate to pocket

Click Apply, Start to pocket holes on targeted plate, cut features apperars of Pocket appears on the property tree.



7. Ejector Manager

Ejector Manager is used to add, edit, and adjust the ejector pin, and trim the core and plate passing through by the ejector. Ejector pin can be added after the mold base has already been loaded

Click  on the 3DQuickMold Toolbar to start the design of the ejector. The steps are as follows,

Define the position of the ejector pin

Add the ejector pin

Trim the ejector pin

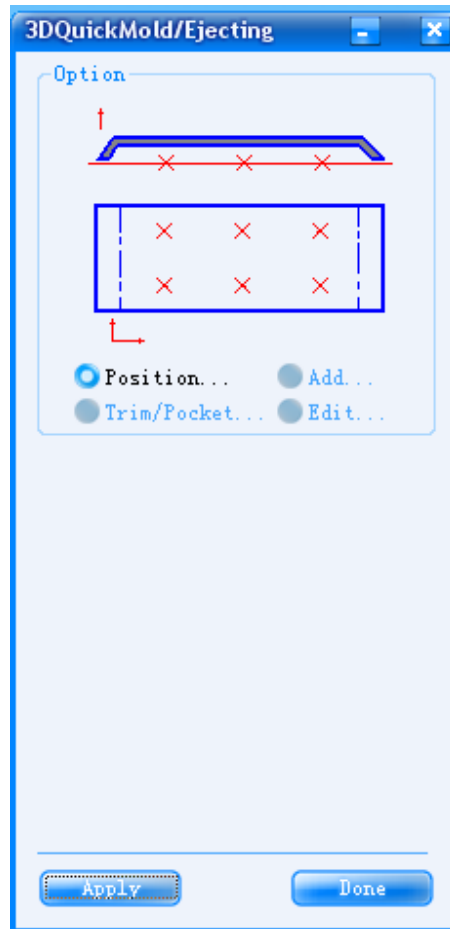
Create pockets on the core and mold plates that the ejector pins are passing through.

Adjust ejector pin if it is not suitable.

1.Position

If the file is a part file, click Ejection Manager, only Position is activated, which is used to define the position of the ejector pins.

There is another method. Use the sketch points on the part to define a 2D sketch, and then define the position of the ejector pins by the 2D sketch.



Click **Apply** to set the position of the ejector pin

Actually, the sketch points for ejector's position could be created on the Core or Product Assembly file. For example, the ejector that is used to eject the runner and gate may not locate on the core or Product, it is usually created on the Product Assembly.

The system advises the user to create the sketch on Product file.

Position Ejectors

Position

Face<1>

☐ Screen point
☐ Middle point
☐ Circle center

X: -23.90mm

Y: -21.60mm

☒ Round up

*

Next

Pattern

2.00mm

☐ Linear pattern
☐ Circular pattern

1

1

D1 10.00mm

D2 10.00mm

1

Select point, edge or face to define the center of the ejector pin here. The center can be defined by the 3 methods below.

Screen point: select the point on the screen;

Middle point: select the middle point of the edge or face;

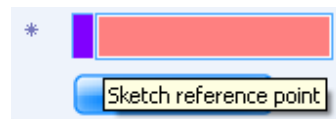
Circle point: Select the center of circle

x, y coordinate system can also be used to define the position (origin as the default reference point)

Round up: Check this option, the system will round up the coordinate value to 2 decimal places.

X:	-46.72605543mm	X:	-46.70mm
Y:	-1.59313743mm	Y:	-1.50mm
<input type="checkbox"/> Round up		<input checked="" type="checkbox"/> Round up	

The default reference point of the x, y coordinate is the part origin.
The reference point can be defined in the Sketch reference point

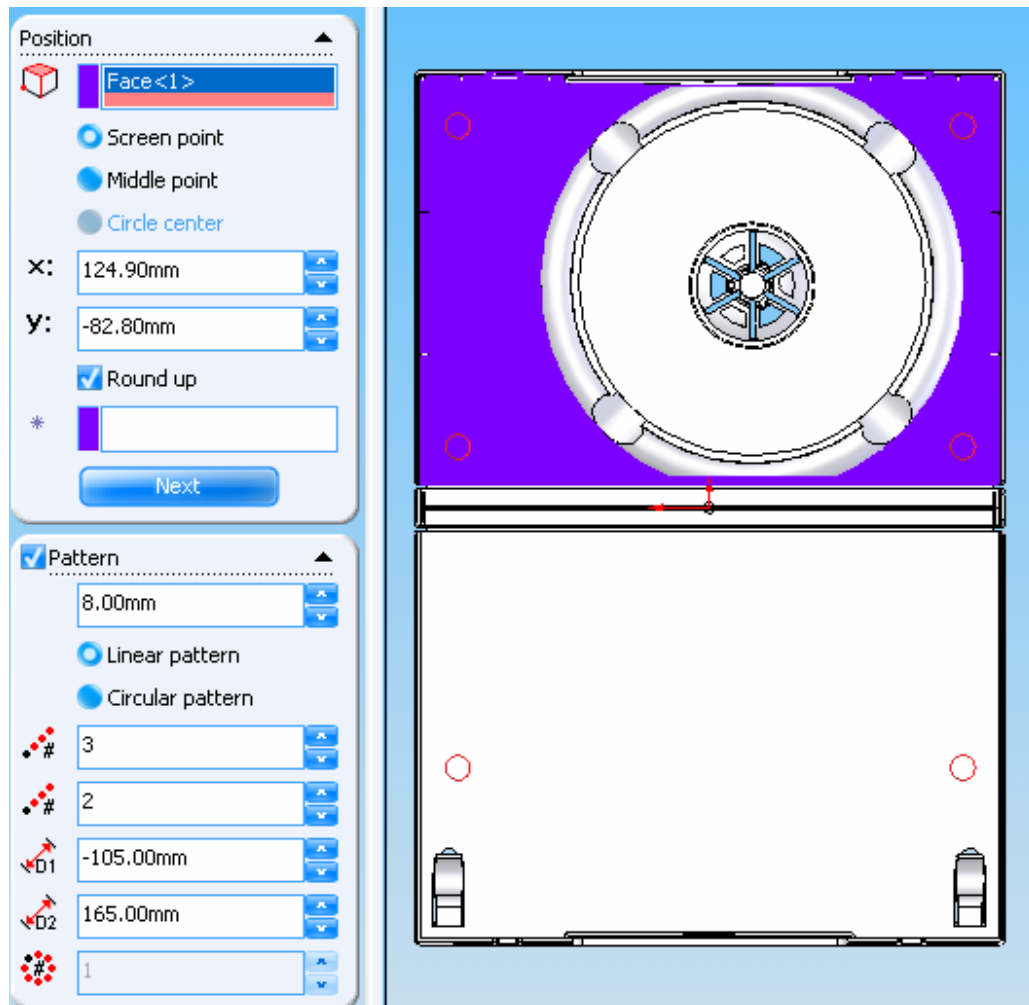


When the position of the point is defined
Click **Next** to create a sketch point and clear the previous ejector location
A new ejector location can be added.

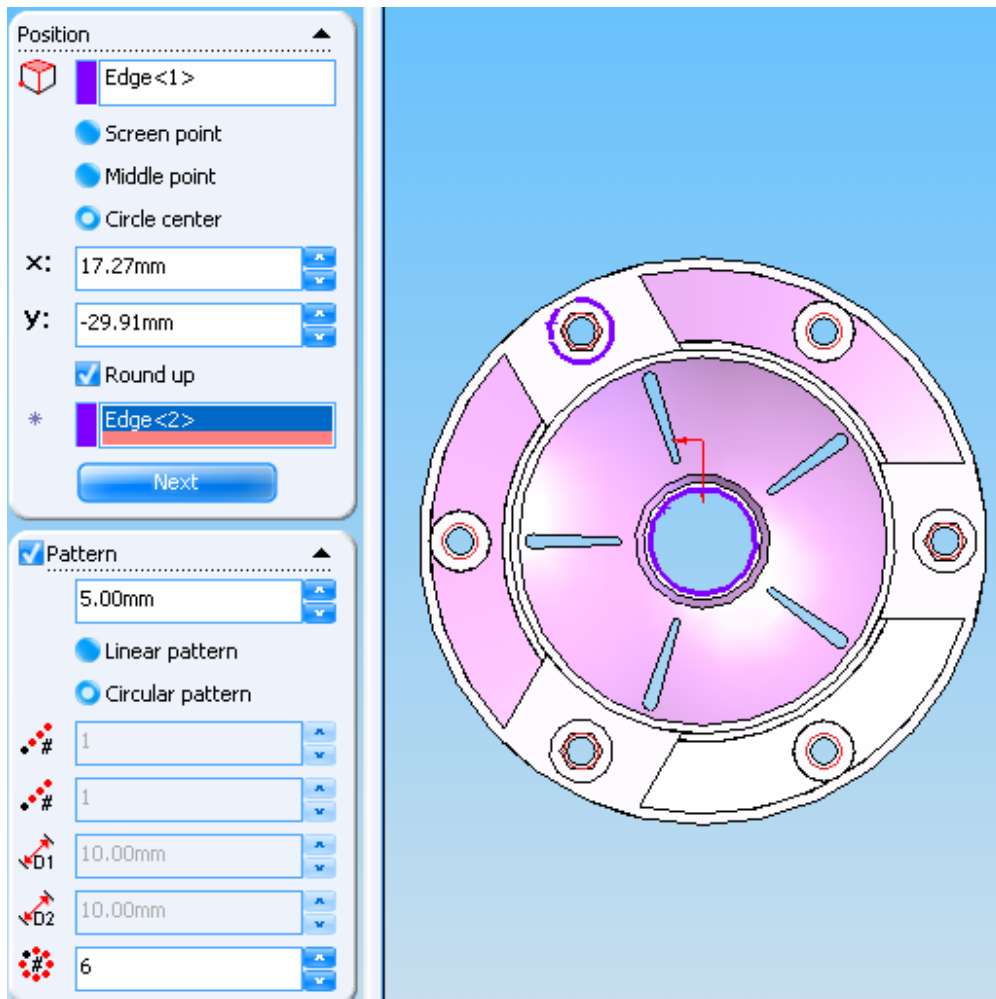
Besides, the pattern of the ejector location can set here.

Ejector Diameter: define the diameter of the ejector, the preview is shown

Linear pattern: generate linear pattern for the current selected ejector location;



Circular pattern: generate circular pattern for the current selected ejector location,



Different data row will be activated for different pattern. Click **Next** to apply.

After setting all the ejector locations, click OK, a new sketch is generated in the feature manager tree.

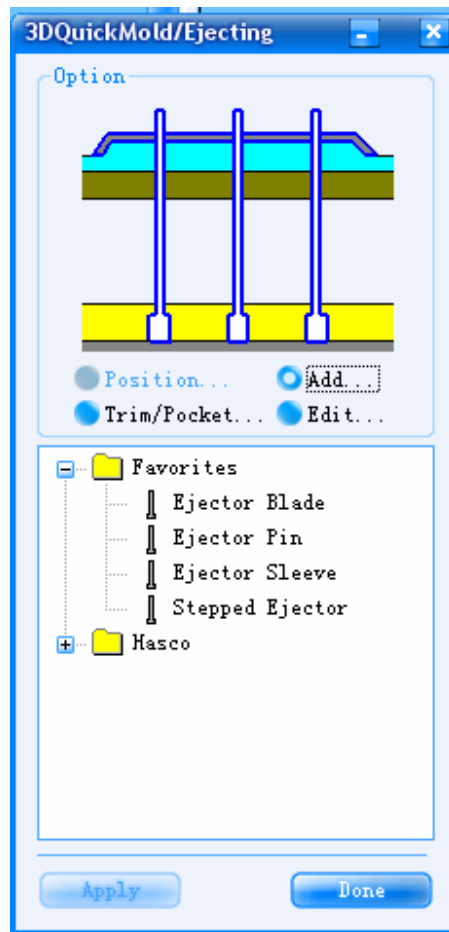
Click Ejector Manager/ Position menu again, new location can be added, the newly add location will be stored in the previous sketch.

The ejector location can be defined outside the **Ejector Manager** by editing the sketch directly.

After the sketch is completed, go back to the * assembly.sldasm(*is the name of the file)to continue to load the ejector into the assembly;

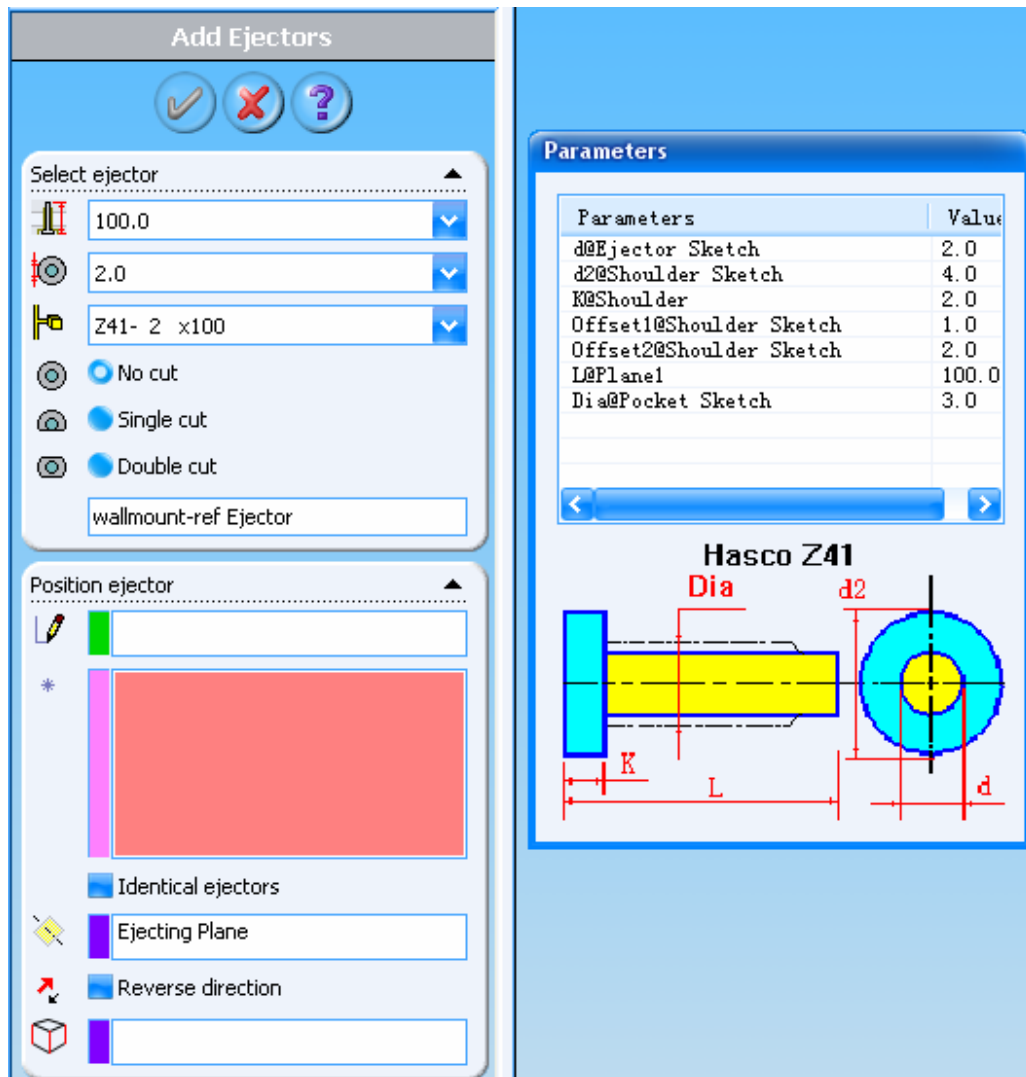
Click Ejector Manager, if the current window is a assembly file, **Position** is disabled.

2. Add ejectors



Add the ejector to the assembly, generate the ejector part and save it to the the working directory.

Add ejector is only for assembly file. Click Add ejectors; select a type of ejector in the library. Click Apply, the Add Ejectors property manager and the Parameters message box pop out.



In parameter, the dimension of each ejector can be edited with reference to the picture at the bottom of the message box. For example, to define the clearance between the ejector and the mold plate, edit Dia and d as shown in above.

Length: select the length of the ejector. The length of the ejector is associated with the configuration. If the length is changed, the configuration will change accordingly;

Diameter: select the diameter of the ejector. The diameter of the ejector is associated with the configuration. If the diameter is changed, the configuration will change accordingly;

Configuration: Select configuration for the ejectors. If the Configuration is changed, the length and the diameter will change accordingly.

No cut: no cut on the ejector flange;


Single cut: Cut the ejector flange on single side;

Double cut: Cut the ejector flange on double sides;

Naming ejector: name the ejector;

In Position ejector

Sketch: Select the sketch generated in **Position**, all the selected ejector locations will be listed in sketch point list, this is a quick selection of all ejector locations.

: Select the location to add the ejector,

If the sketch is selected in the previous step, the points in the sketch will be added automatically to here;

the points of ejectors of the same type and same configuration can be selected.

To select the points of ejectors of different type or ejectors of same types but with different dimensions require more steps.

Identical ejectors: select whether to add the same ejectors for the above selected points. It is especially for interchanging ejector, reduces the number of different parts in the assembly, as a result easier to manage.

Reference plane: the reference plane which the ejector will mate to.

In 3DQuickMold, the Ejecting plane is the default reference plane. Other plane such as Bottom plane will be used to place ejector sometimes.

Reverse direction: reverse the direction of the ejector;

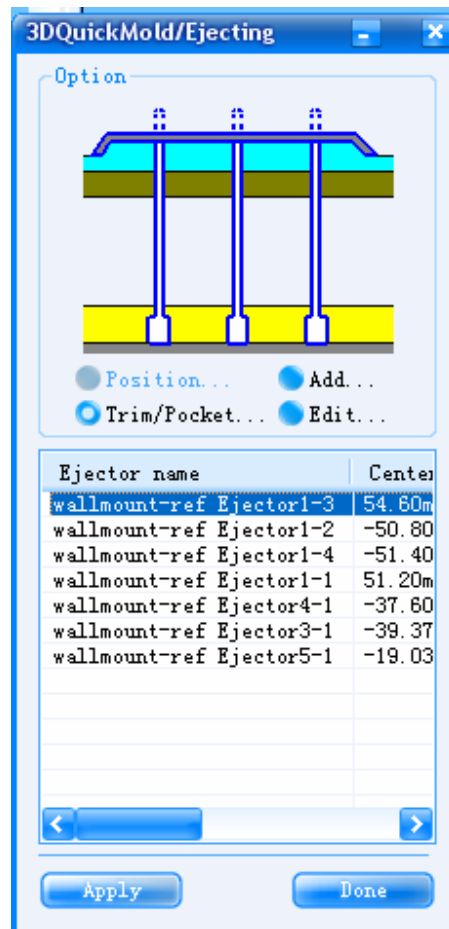
Select a circle: It can determine the diameter of an arc. Select an arc, a message box showing the diameter of the arc will appear at the lower left hand corner. It helps to determine the size of ejector easier. Particularly, for ejector sleeve.

After the setting is done, click the OK button, 3DQuickMold will load the ejectors automatically, the ejector generated will be saved in the current working directory, it take a moment to generate all the ejector if the no. of ejector is too many;

In addition,

- When the ejector is added, it is not yet fully constrained. Still can rotate, for ejector blade, additional mate is necessary to align its orientation.
- Ejector sleeve will take two steps to complete the design. Put an ejector on the Bottom Plane and Ejector sleeve on the Ejector Plane.

3. *Trim/Pocket*

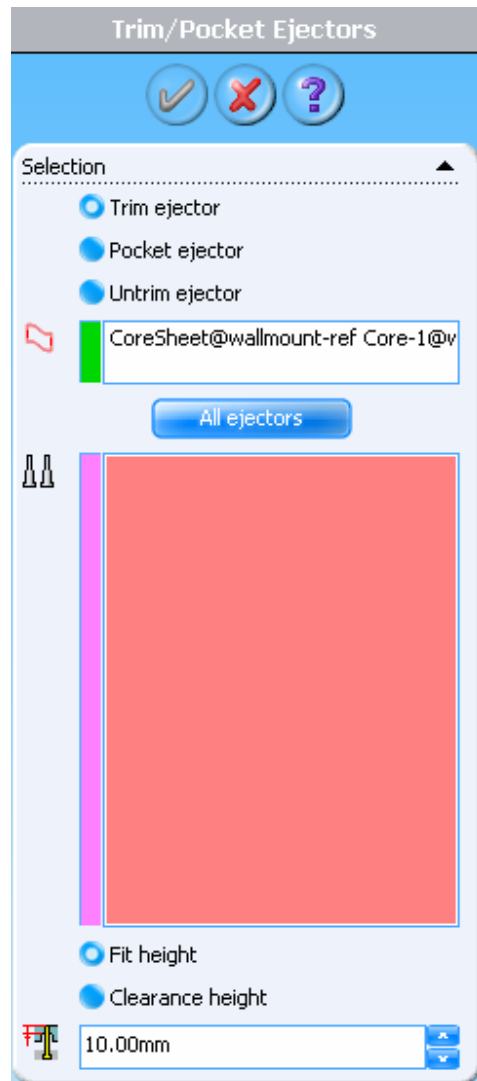


This function is mainly used for

1. Trim ejector
2. Create pockets for ejectors
3. Undo the trimming

To trim the ejectors automatically, there must be a sheet body named as CoreSheet in the core part which is actually the entire core faces.

When the core/cavity is separated successfully, the CoreSheet should be generated.



Trim ejector: trim ejector so that the end surface of the ejector match the core profile.

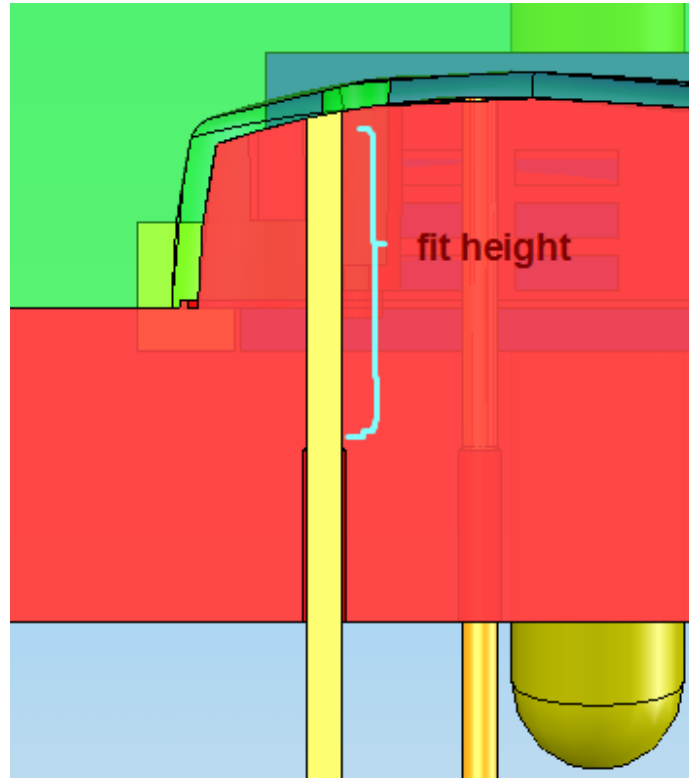
Tool surface: cutting tool surface, use this surface to trim all the ejector end surface, as default, 3DQuickMold will select the CoreSheet in the * core (* represent the file name of the plastic part) part for the tool surface, user can define their own tool surface to trim the ejectors;

All ejectors: Select all ejectors

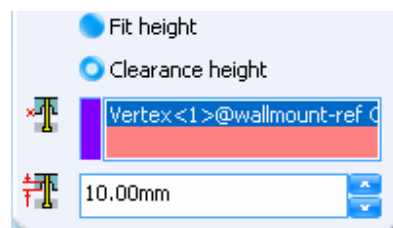
Select ejector: Select the ejector to trim;

Fit height: the height the ejector fit with the core.

Value of the fit height: select the fit height of the ejector pocket on the core insert. The value is measured down from Tool surface.



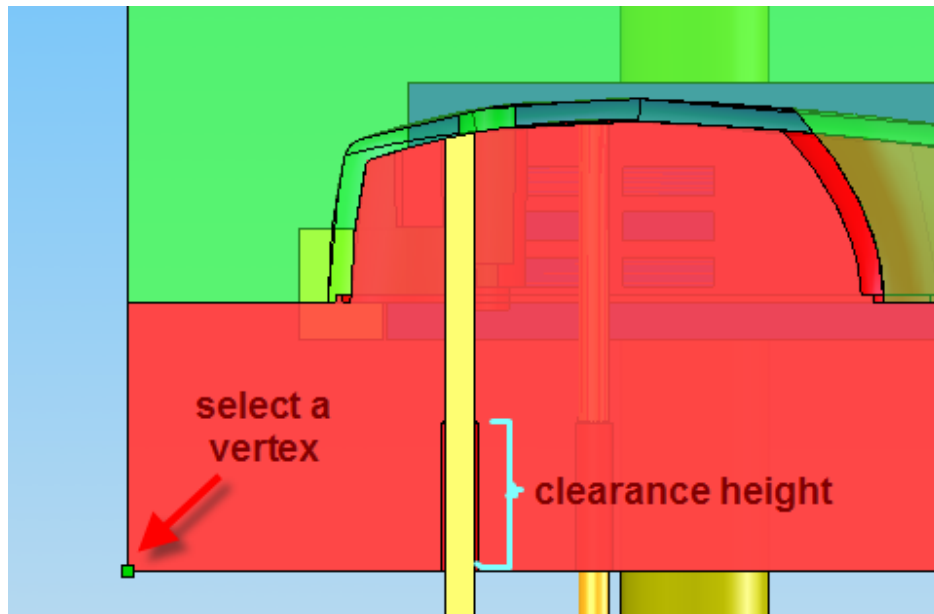
Clearance height: select this will influence the pocketing behaviour of the core insert, it is the height of the clearance pocket measured up from the bottom surface of the core plate or from a vertex selected;

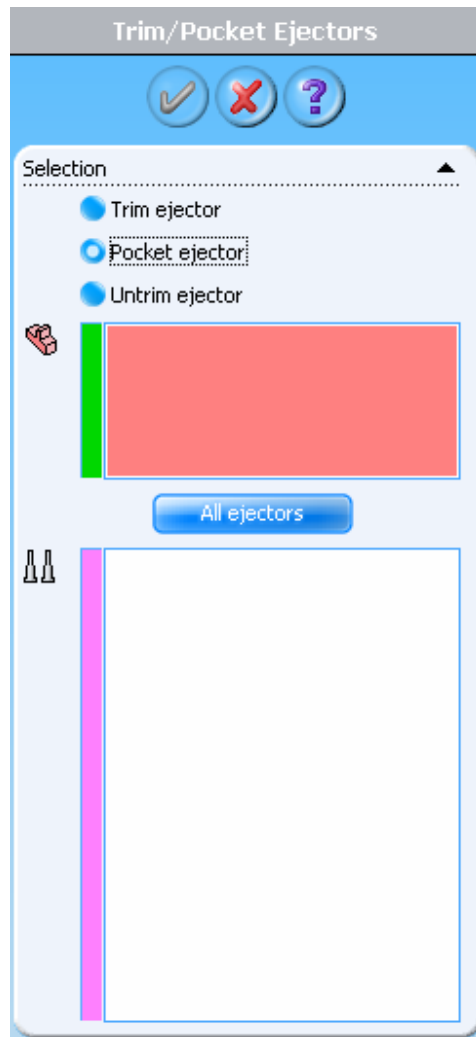


Vertex: select a vertex as the starting point to measure the clearance;

Clearance: select the clearance height. The value is measured from the vertex in downward direction.

After the setting is completed, click OK, 3DQuickMold will trim the ejector automatically, the trimmed ejectors will then be rebuilt. If there are many selected ejectors, it may take a longer time to trim.





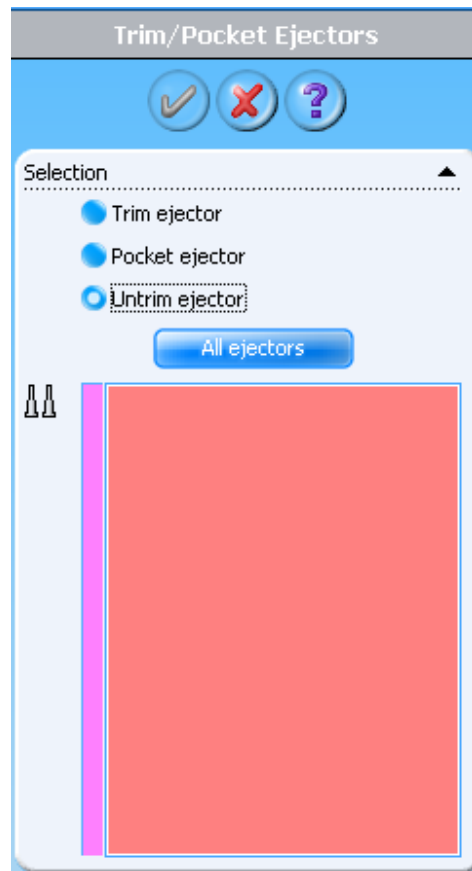
Pocket ejector: Pocket the mold core and mold plate which the ejector passing through.

Tool component: Here are the ejectors to create the pockets

All ejectors: select the ejectors under the current assembly

Select ejector: select the ejector as pocketing tool;

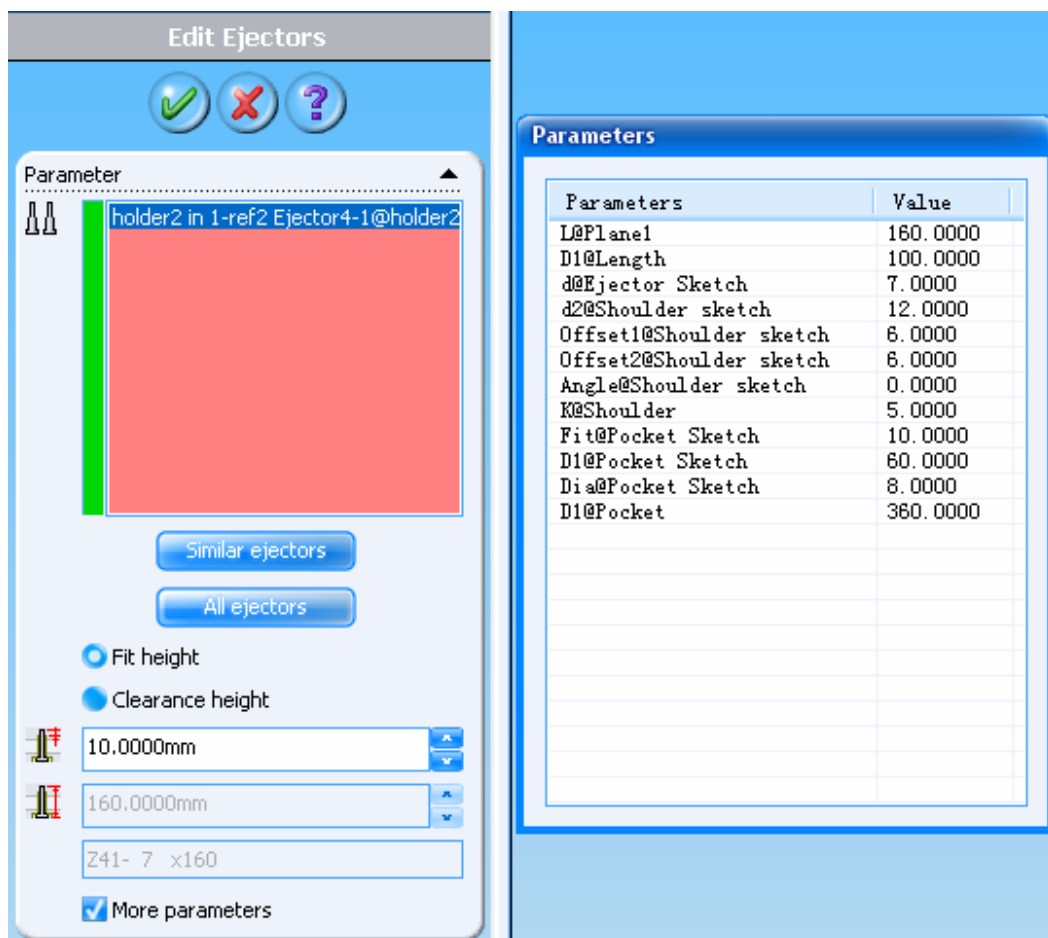
After set up is completed, click OK, 3DQuickMold will pocket the selected mold component, the pocketed component will be rebuilt.



Untrim ejector: Restore the trimmed ejectors. When performing Trimming, if there is error on selecting the CoreSheet or the Core requires a great change, use this function to restore the ejector.

4. *Edit Ejectors*

To change the Fit height and Clearance height of ejector and core. Different parameter can be changed, especially the clearance.



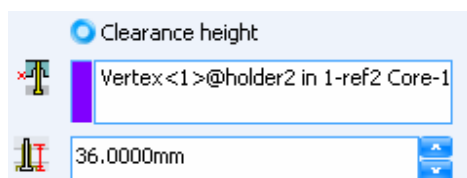
Select ejector: select the ejector to edit;

Similar ejectors: select all the ejectors of the same type and same configuration

All ejectors: select all ejectors and add to the ejector selection list

Fit height: After selection, the fit height of the ejector and Core is displayed below

Clearance height: After selection, the Clearance height of the ejector and Core is displayed below



Original length: total length of the ejector. It is for reference only, it cannot be edited.

Ejectors type: Ejector type. It is for reference only, it cannot be edited


More parameters: After selection, the following dialogue box appears, it displays all the details of the selected ejectors.

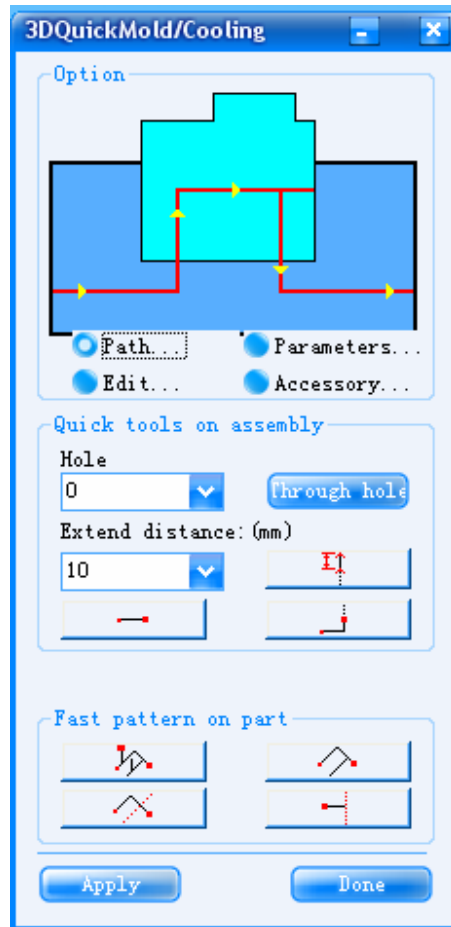
8. Cooling Manager

The uses of Cooling Manager are to build, edit water channel, and add common standard parts of cooling to the mold. Normally, the channels are built on the core plate, cavity plate, sidecore and larger insert.

The normal design procedure:

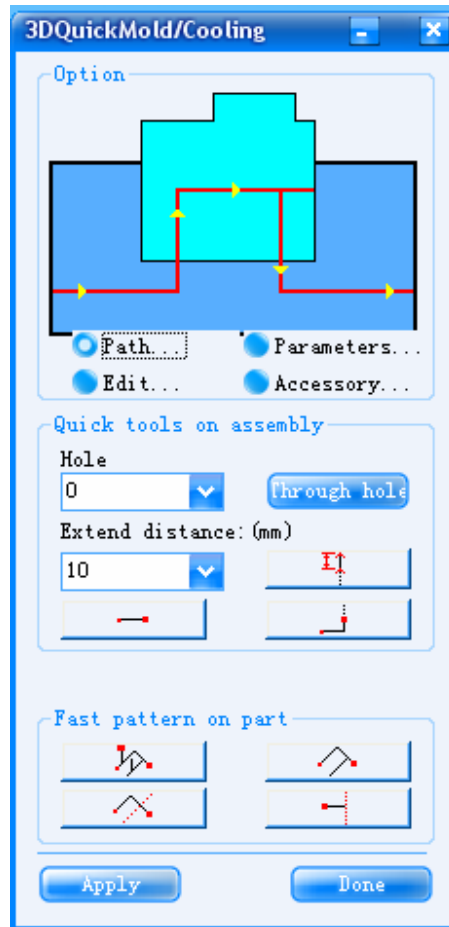
1. Quickly define pattern and path of the cooling channel, cooling channel is a 3D sketch
2. Define the accurate position of the sketch
3. Set the parameters of the cooling channels
4. Sdd standard cooling accessories.

Click  and the dialogue box pops out



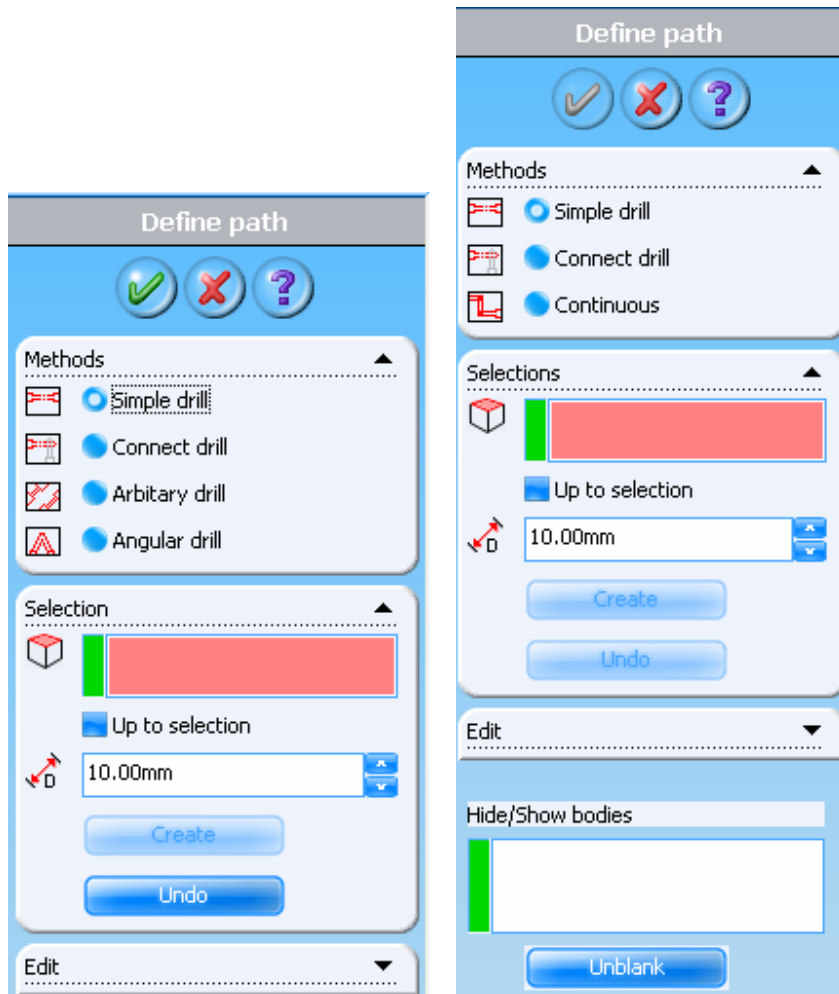
All the design process of cooling can be completed in this module. Four options are provided which are path、parameters、edit and accessory;

1. Path



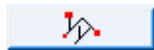
3DQuickMold provides the following design methods of the water channels.

1. Quick tools on assembly: Apply on assembly file, it is recommended to apply this method on the mold plate.
2. Fast pattern on part: Apply on part file, it is recommended to use this method on the mold core and cavity.
3. Click Apply, the box below pop outs. Different boxes appear for part file and assembly file.

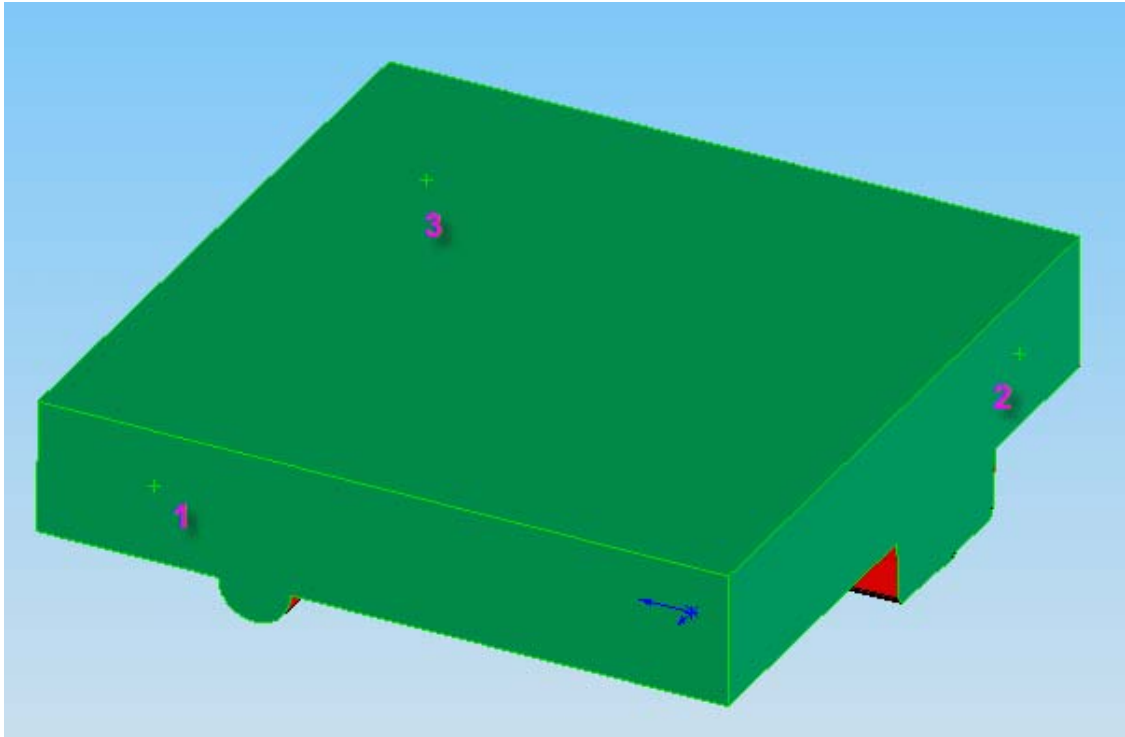


User-defined sketch can be used to create cooling channel, the tools provided are just for convenience.

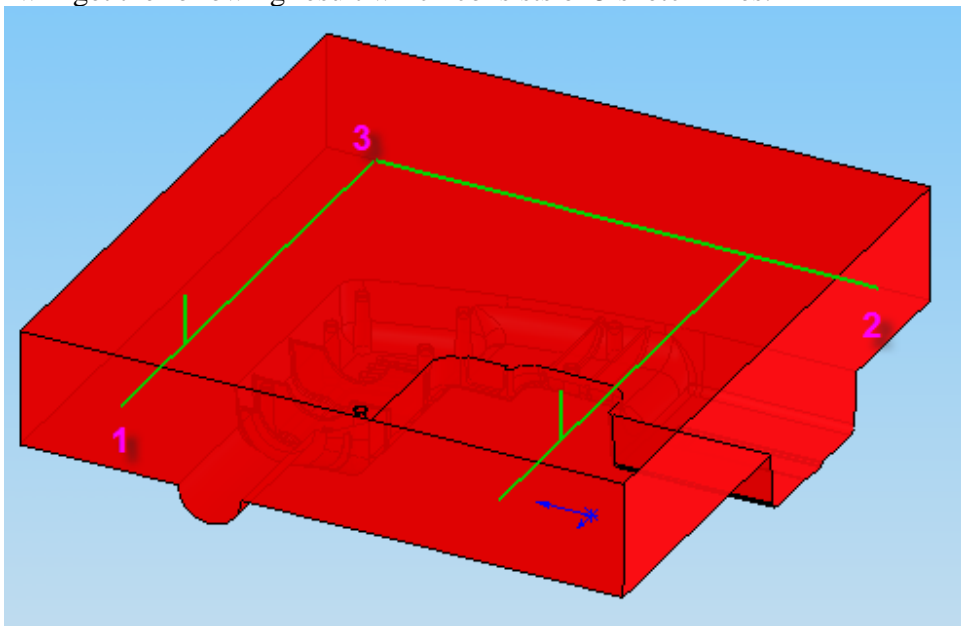
The icons on Fast pattern; The icons are applied on part file, the following demonstrates the function of each icon.



: As shown below, select 3 points in sequence on part (it could be cavity, core or side core), Click this icon.

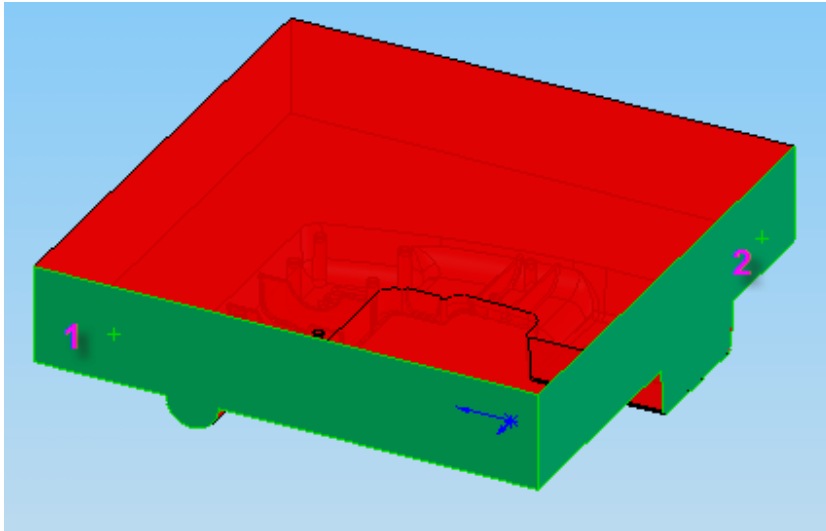


User will get the following result which consists of 5 sketch lines.

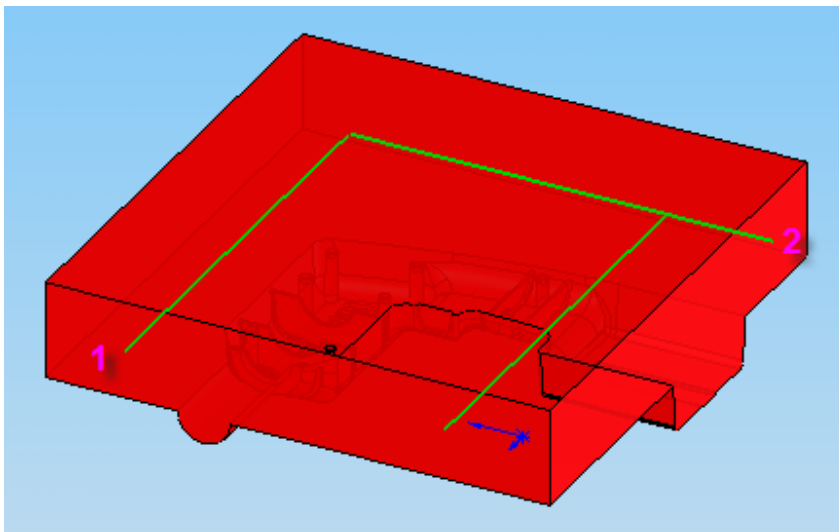




: As shown below, select two points and click this icon.



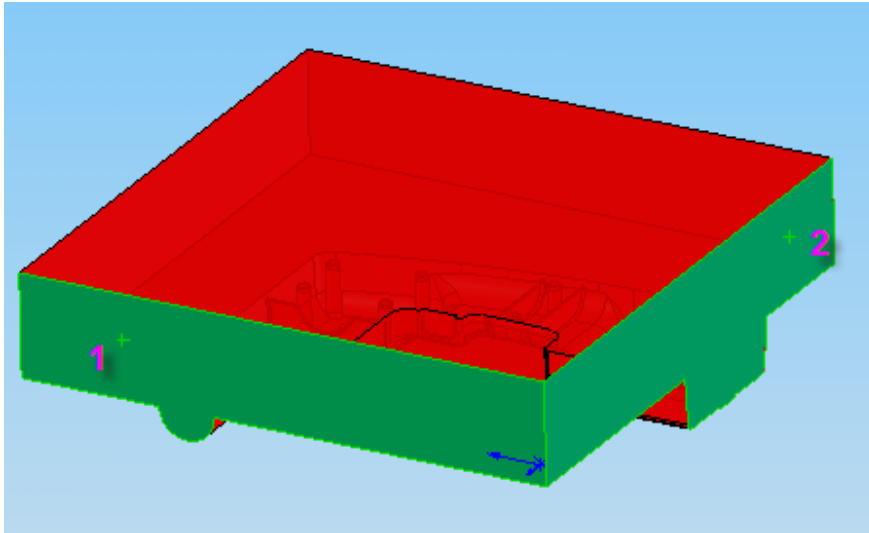
Three sketch lines will be created as below



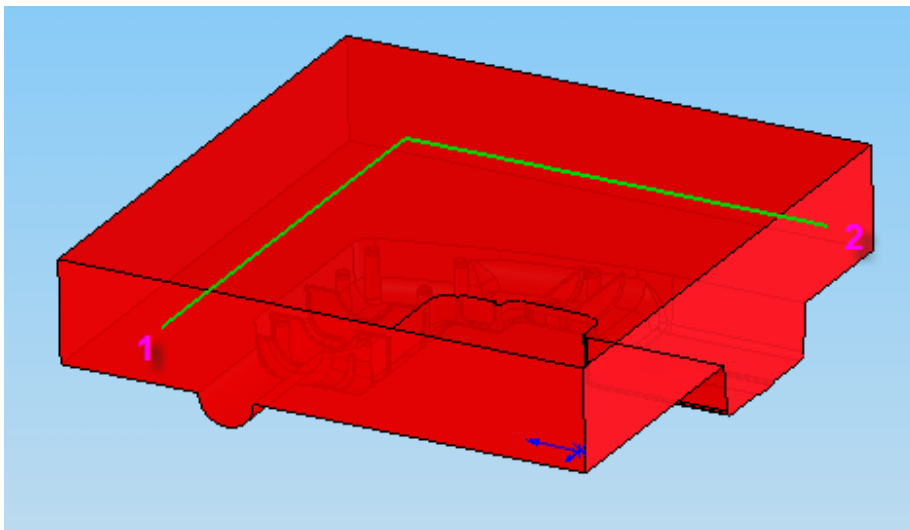


: Below provides two methods of combination.

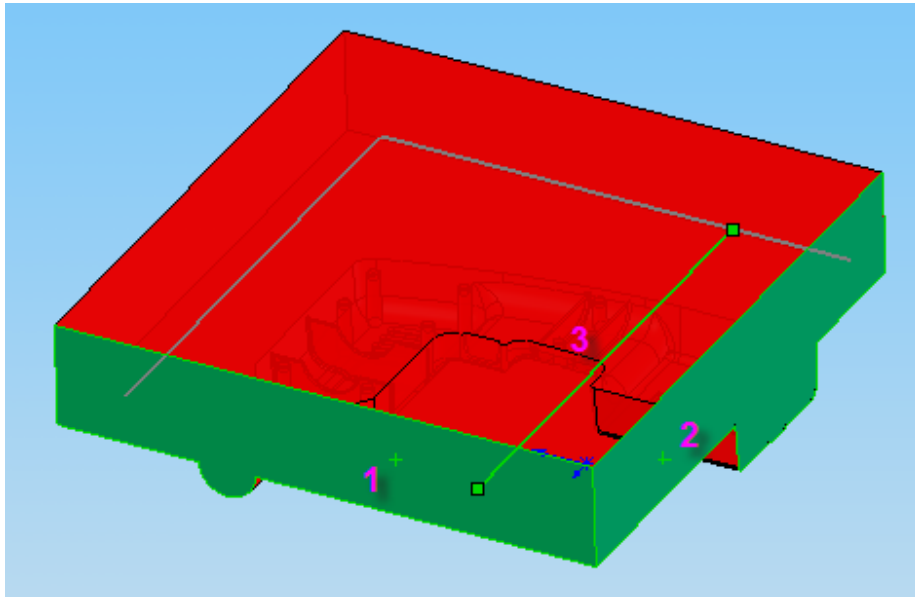
Method 1: Select two points in sequence.



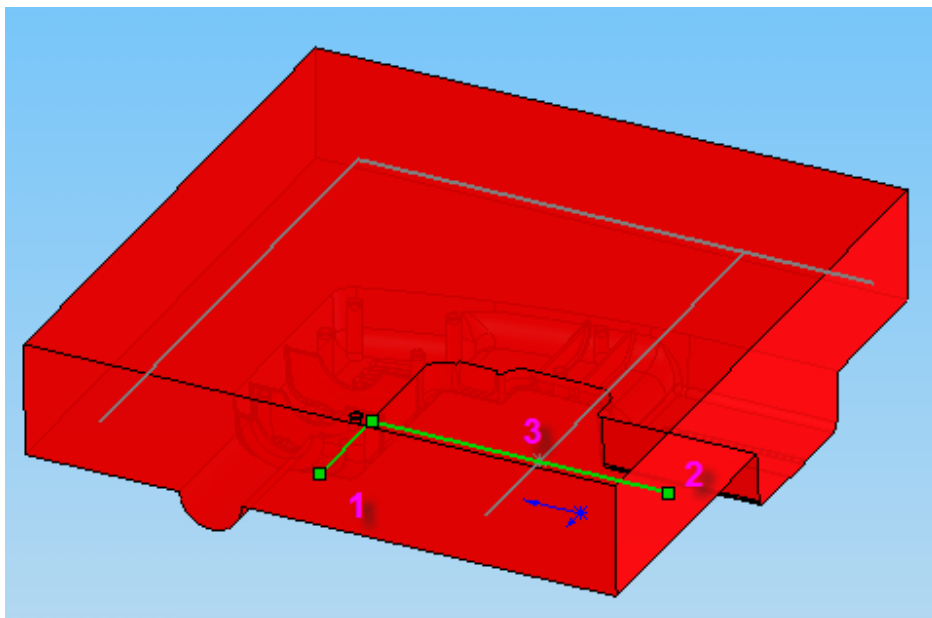
A sketch with two lines.




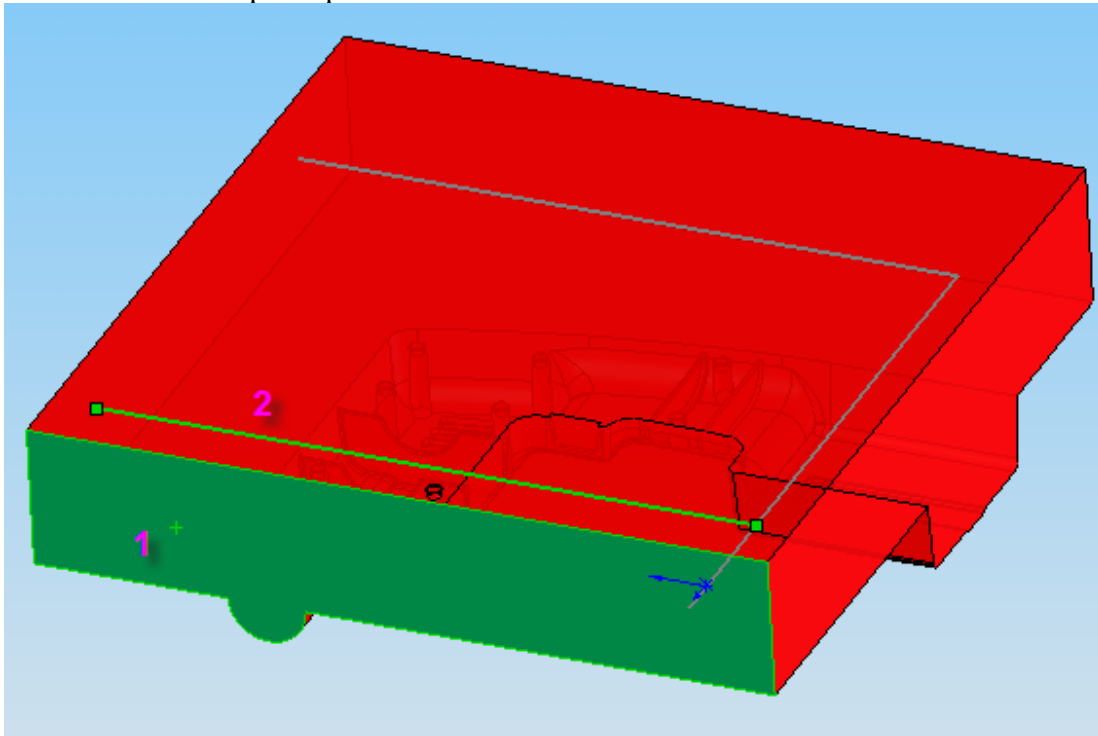
Method 2: Select two points in sequence and select one more sketch line 3 as reference.



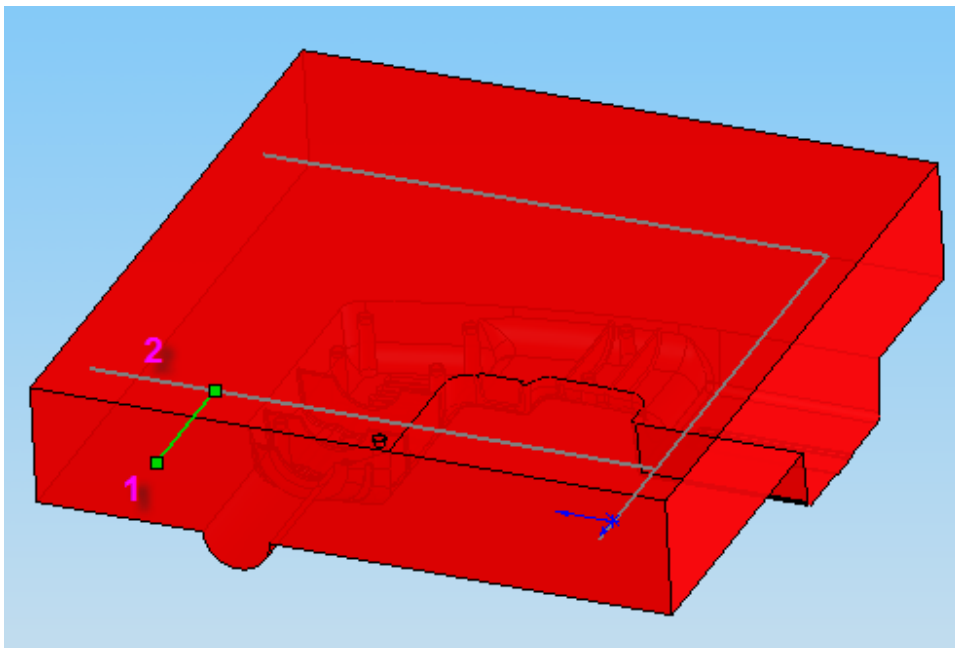
The selected line 3 indicate that the cooling channel passes through the selected reference sketch line



 : Pick up one point and one sketch line



A connected sketch line is create.

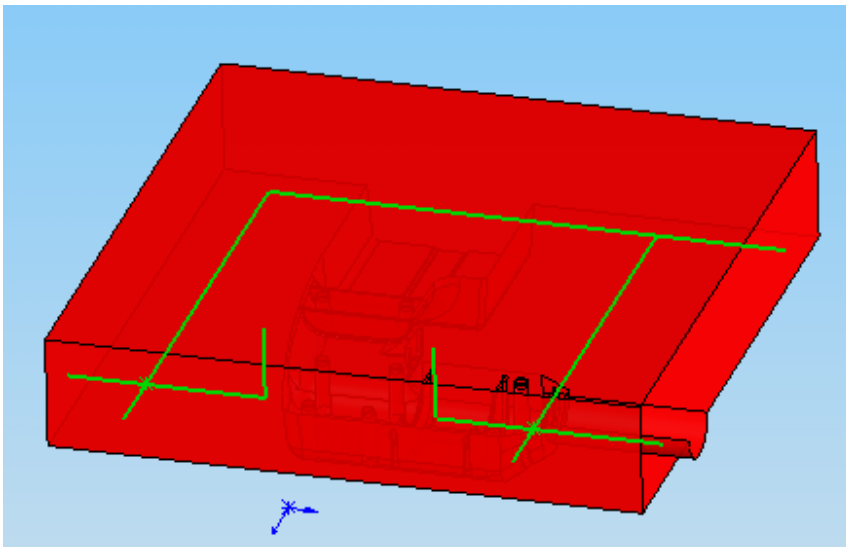



The selected points and line above is corresponding to the points and line in red on the icon and form the water path corresponding to the black line on the icon. Besides, the order of point and line selection affects the feasibility of constructing the channel and the outcome.

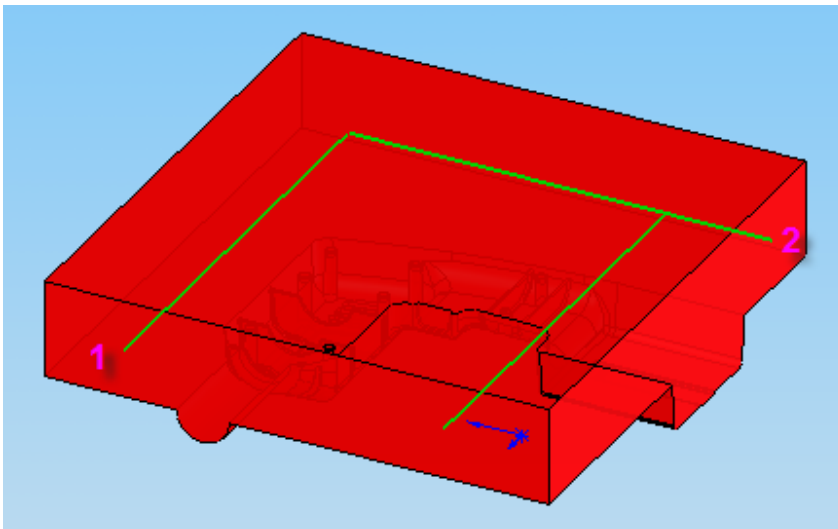
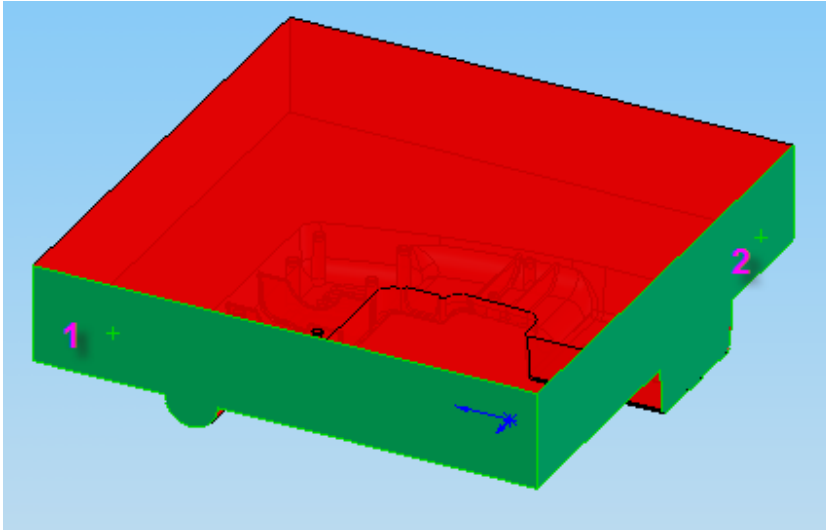
After quick construction of the water channel, a 3D sketch is created in the feature tree. If positioning the cooling path is required, edit the sketch.


Note: the construction of complex cooling path may require several methods.

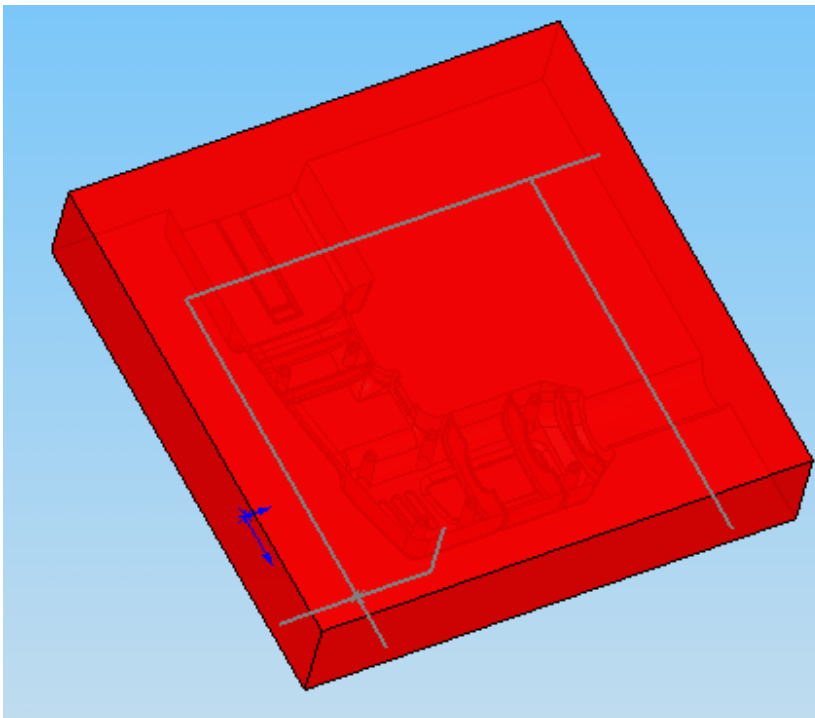
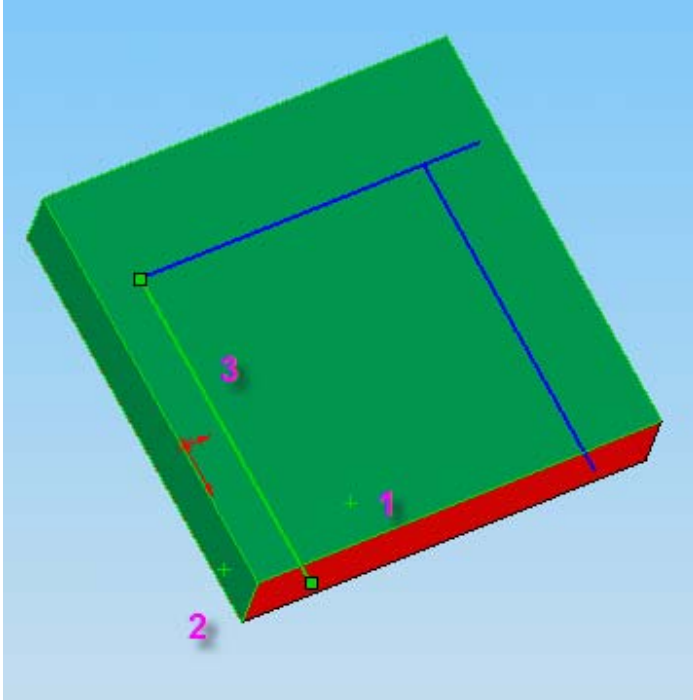
The following example demonstrates a usual method to construct cooling path.

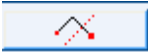


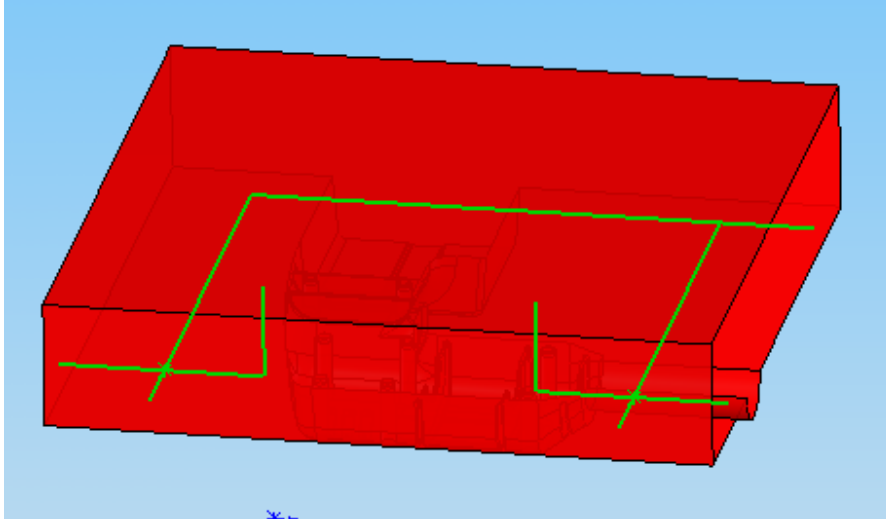
Click  to finish the first three sketch lines.



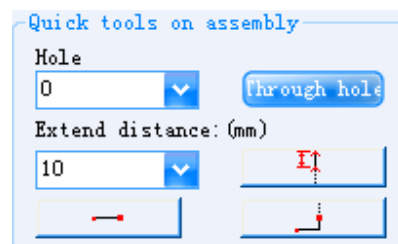
Then click  with two points selected and one sketch line as reference



Click  again to obtain the last two sketch lines



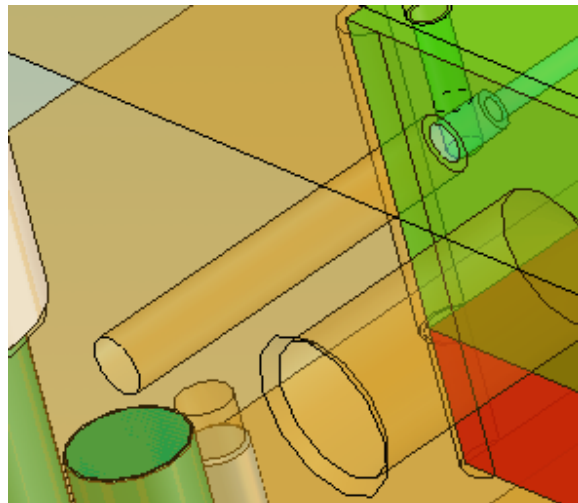
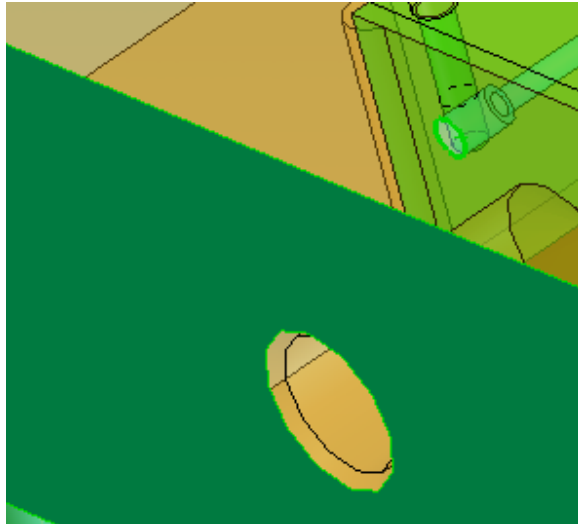
Quick tools on assembly: This function is for the assembly file only. The following demonstrates the use of each command.




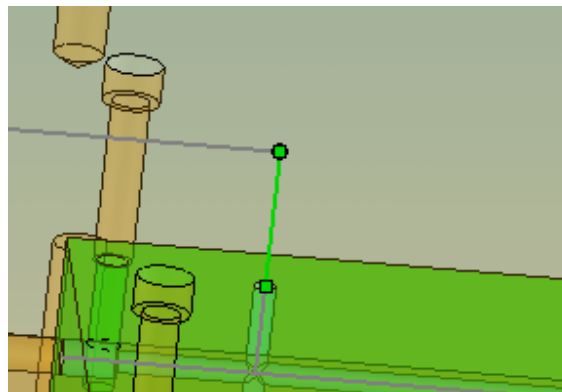
through hole: Cut hole directly through the mold plate

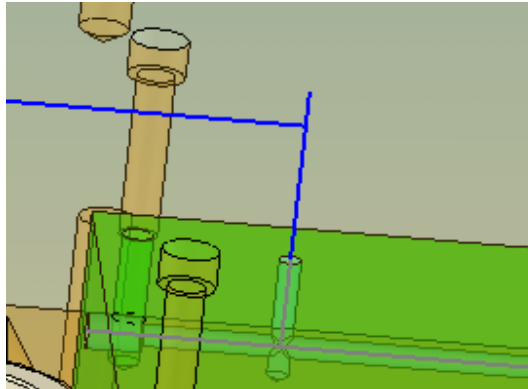
Procedure:


1. define the diameter of the hole . if the input is 0, the diameter will be the diameter of the selected circle.
2. Select in sequence, a face, the circular edge of the hole of the cooling channel.

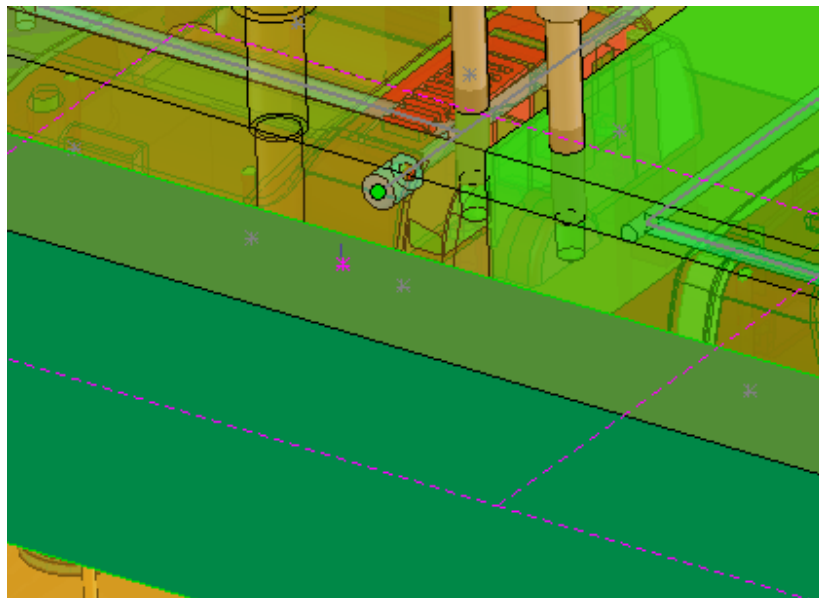


: extend the cooling sketch line, set the length of extension. Select line segments and its vertex to extend respectively.

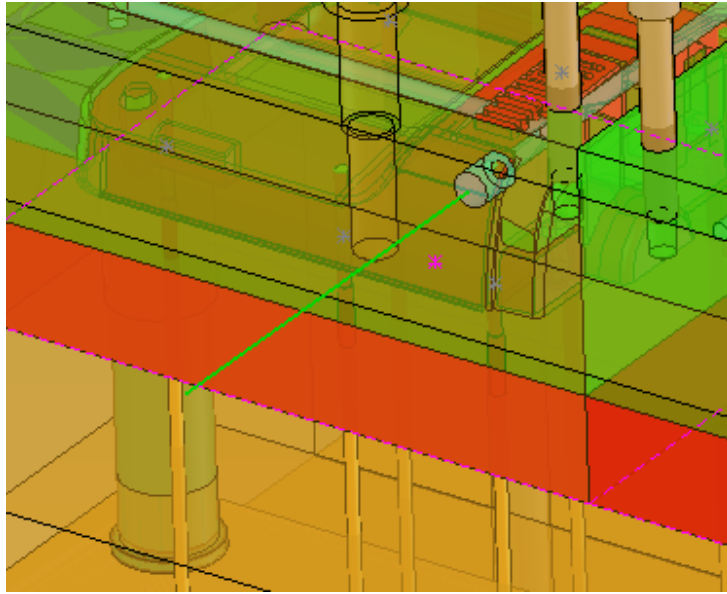




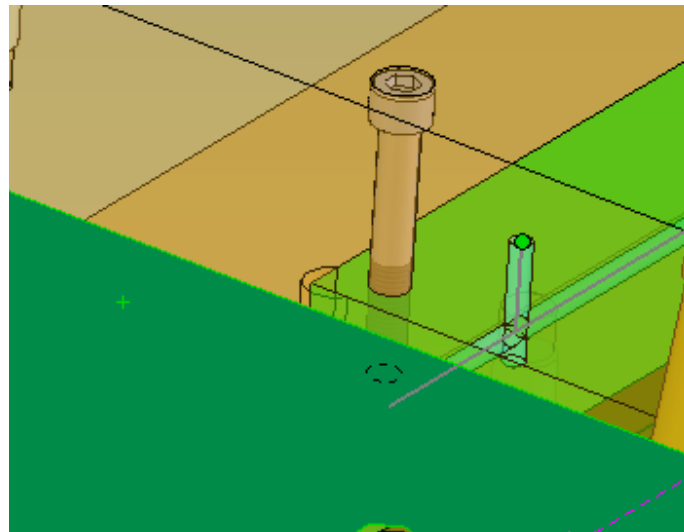
: Select the face and sketch point in sequence.

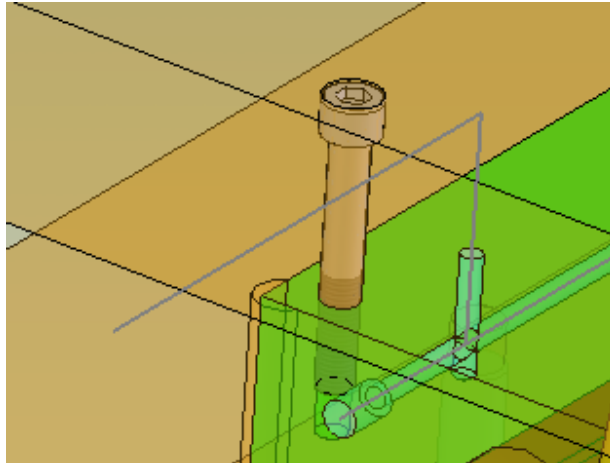


Click this icon 

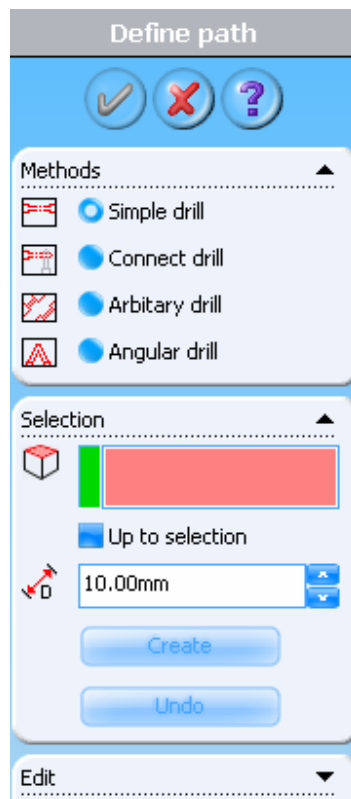


: Select the face and sketch point in sequence.






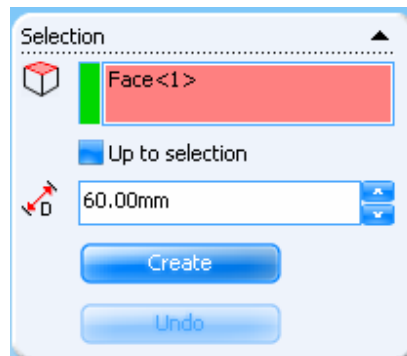
For part file, when Path option is selected, click Apply, the interface pop outs as shown:



Methods available for creating the cooling path:

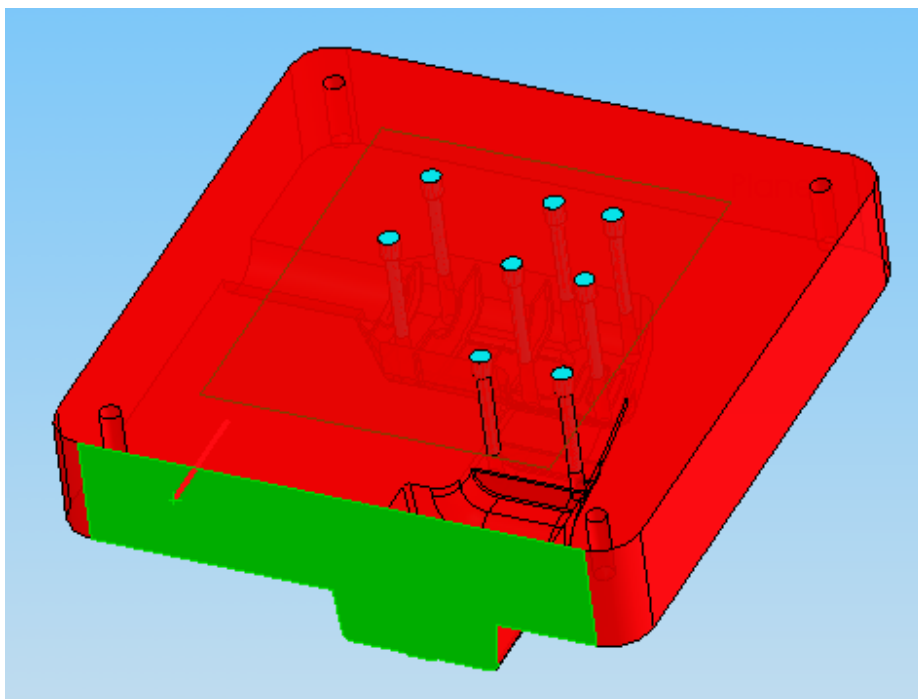
- Simple drill:
- Connect drill:
- Arbitrary drill:
- Angular drill:

 **Simple drill:** The cooling channel built is perpendicular to the selected face and extends to a certain length or passes through a certain face.

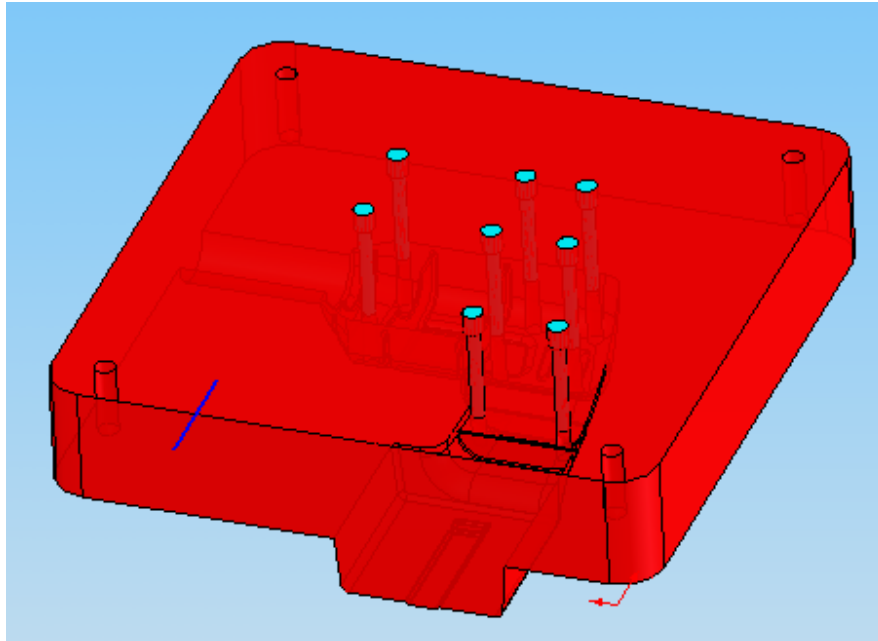


Drill from: Select a face as reference to construct the cooling path. The preview of the sketch line is displayed.

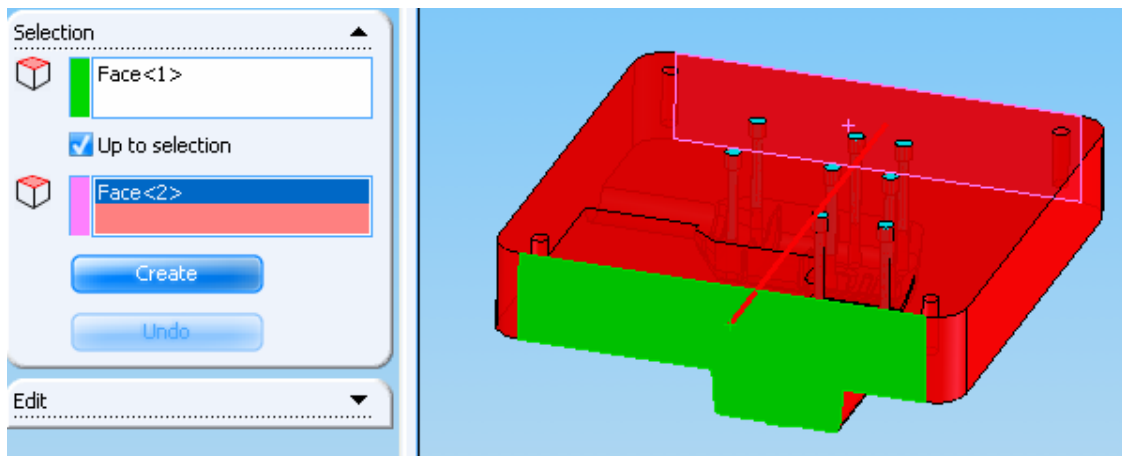
Drill length: Define the length of the sketch line.



Click **Create** to create the sketch. Click Undo to cancel the last created sketch line.

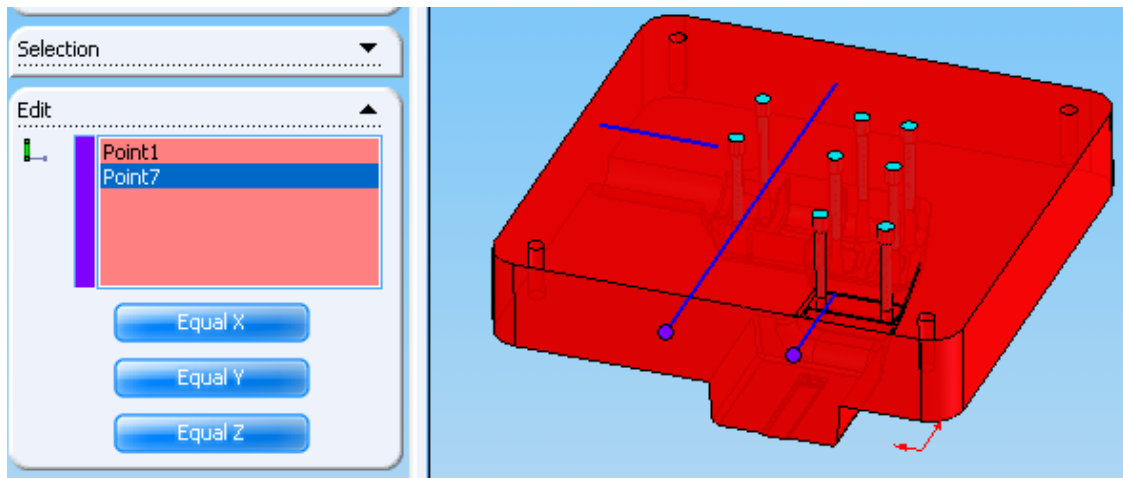


Check **Select Up to selection** to build path between 2 faces.

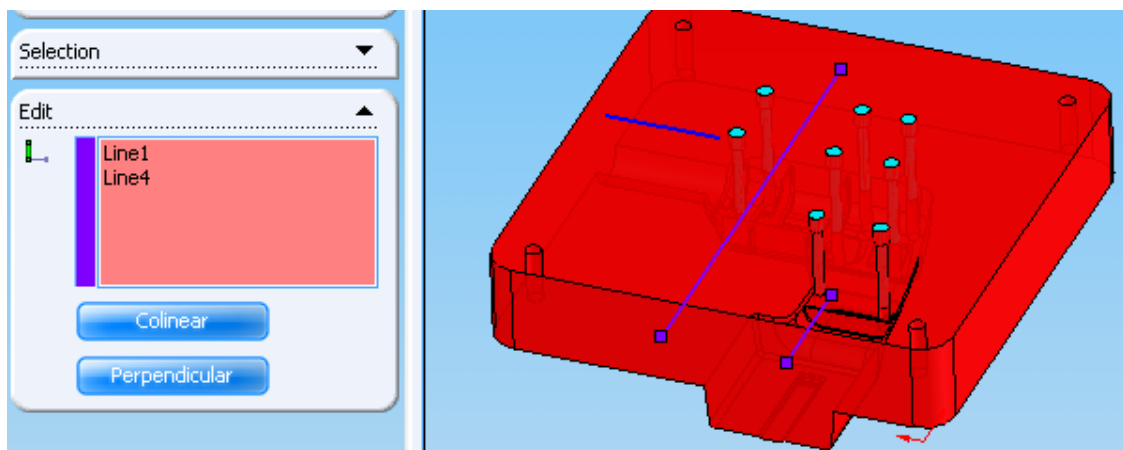


Edit: initially set the position and relation. There are two cases


1. Point and Point: Adjust position between points:
The selected points could be set as Equal X, Equal Y and Equal Z.

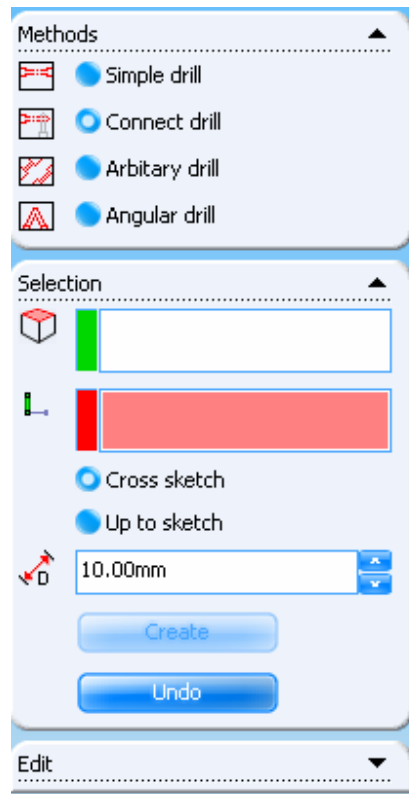


2.Line and Line: Adjust relationship between lines. They could be co-linear and perpendicular.



Rule of adjusting the positions: The first selected line is moved and the latter selected lines will not be moved.

 **Connect drill:** Create a sketch line that is connected to another sketch line

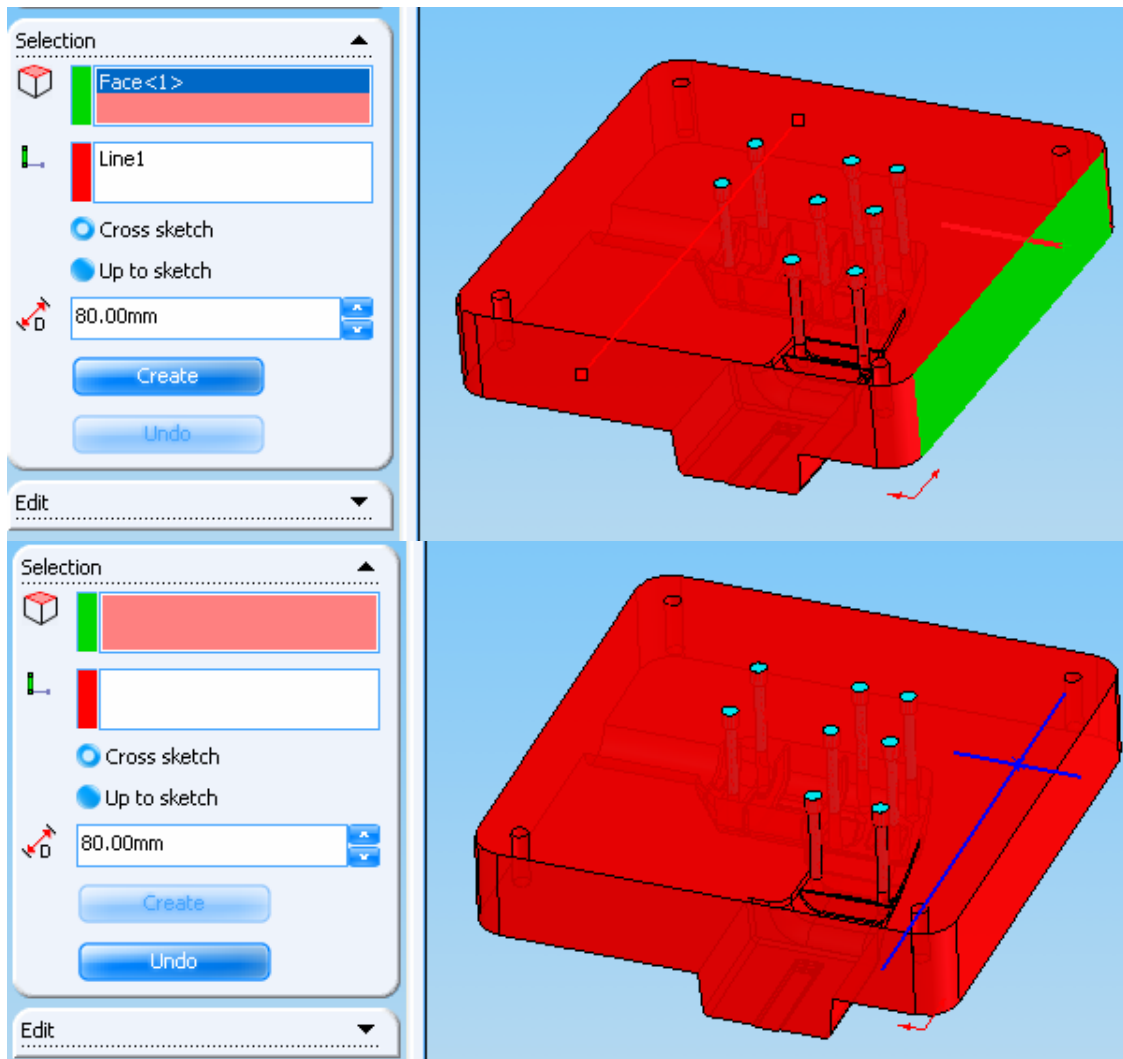


Drill from: Select a face as the starting face of a sketch line, the preview is displayed.

Select sketch segment: Select another sketch line as the connection.

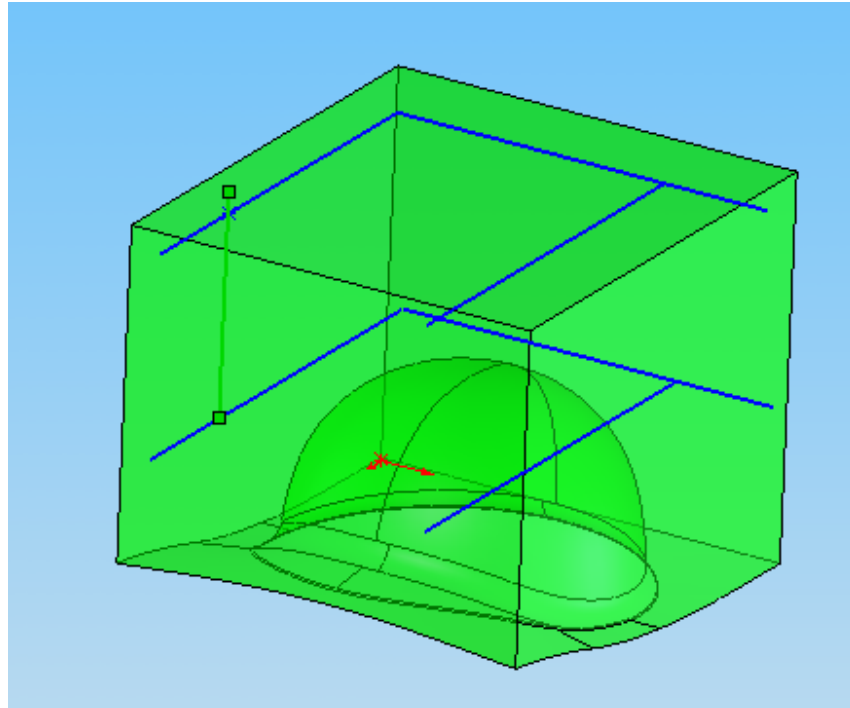
Cross sketch: the newly generated sketch line pass through the connection path, but the length is defined by dimension.

Drill length: Define the length of the sketch line (cooling path).

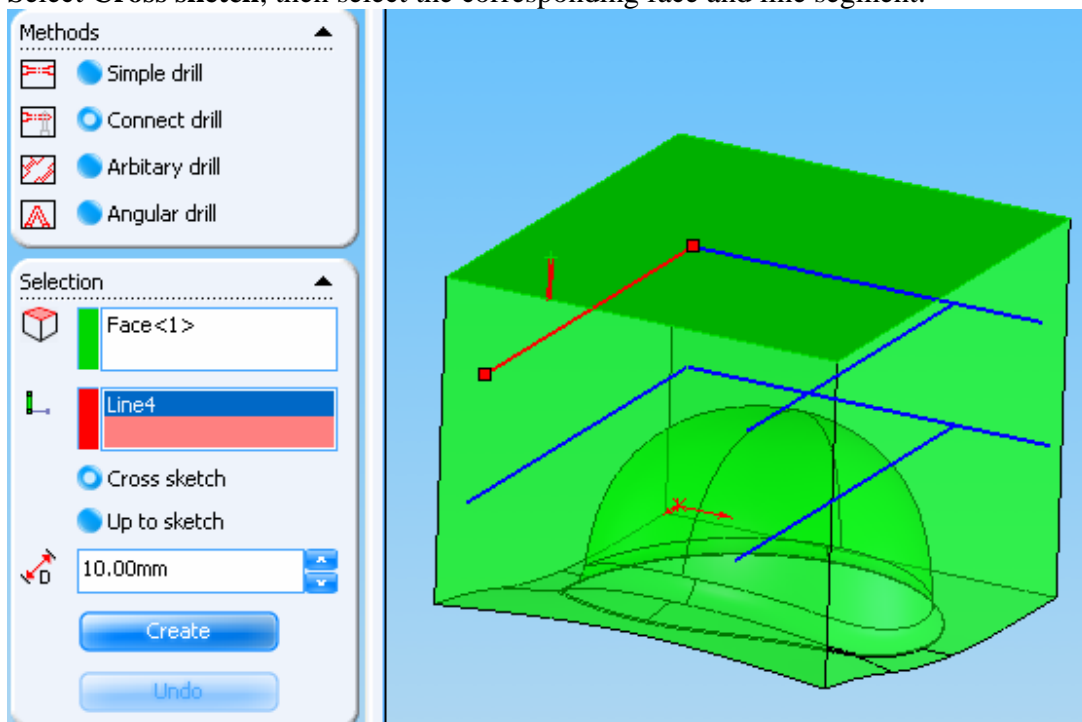


For a water channel with many layers, Cross sketch is a useful tool

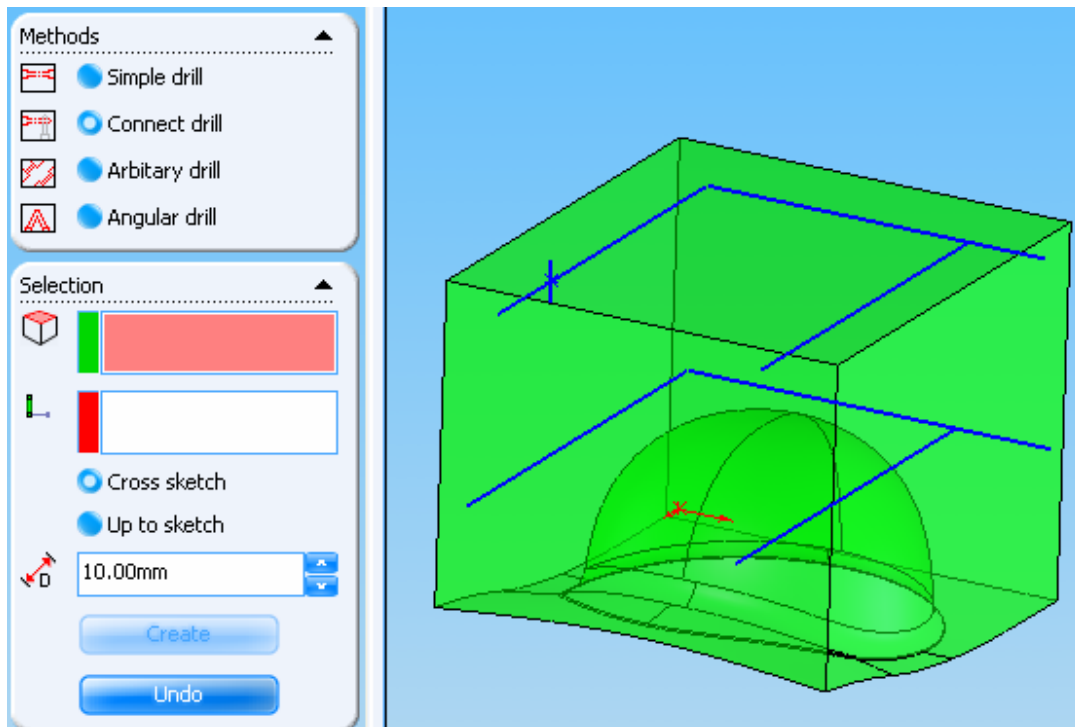
For example: construct the selected vertical path passing through the two layers of horizontal cooling path.



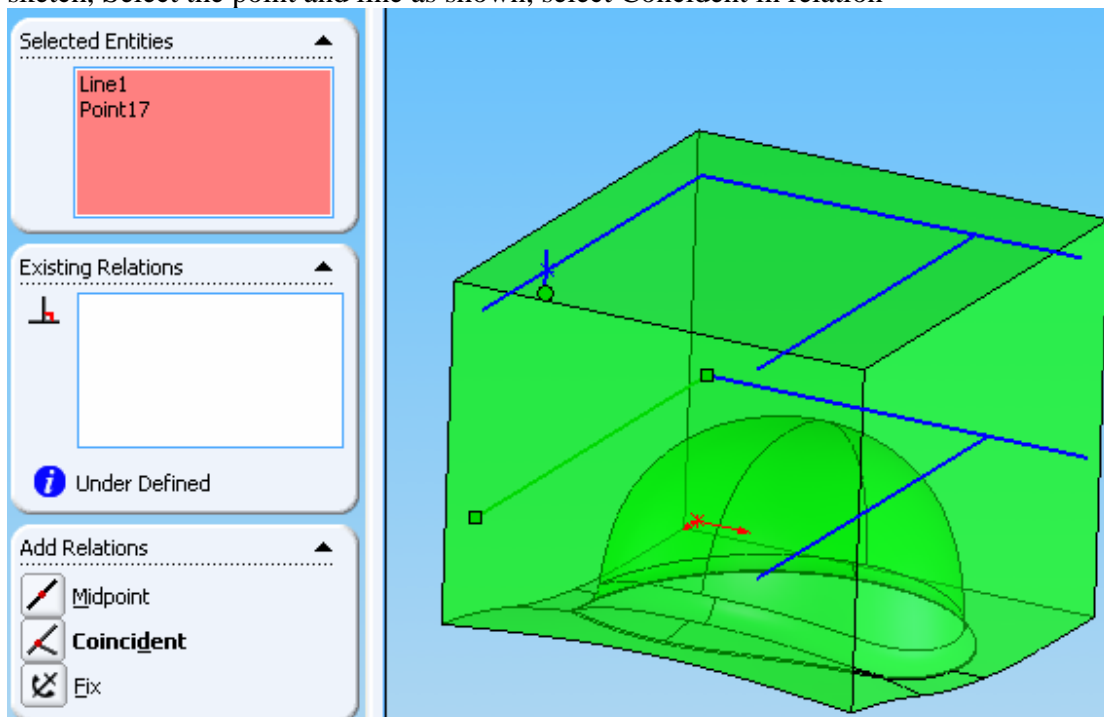
Select **Cross sketch**, then select the corresponding face and line segment.



Click **Create**, the following line segment is obtained.

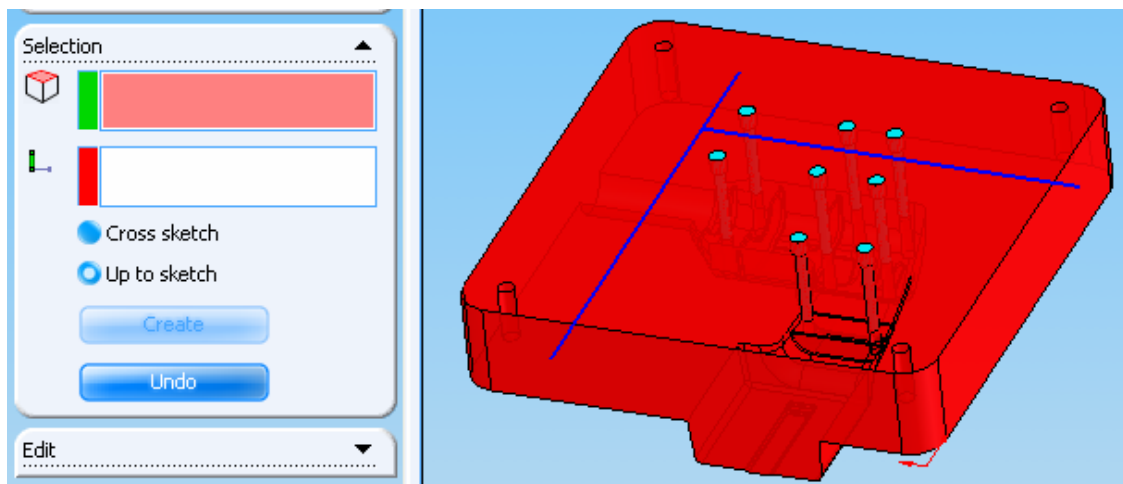
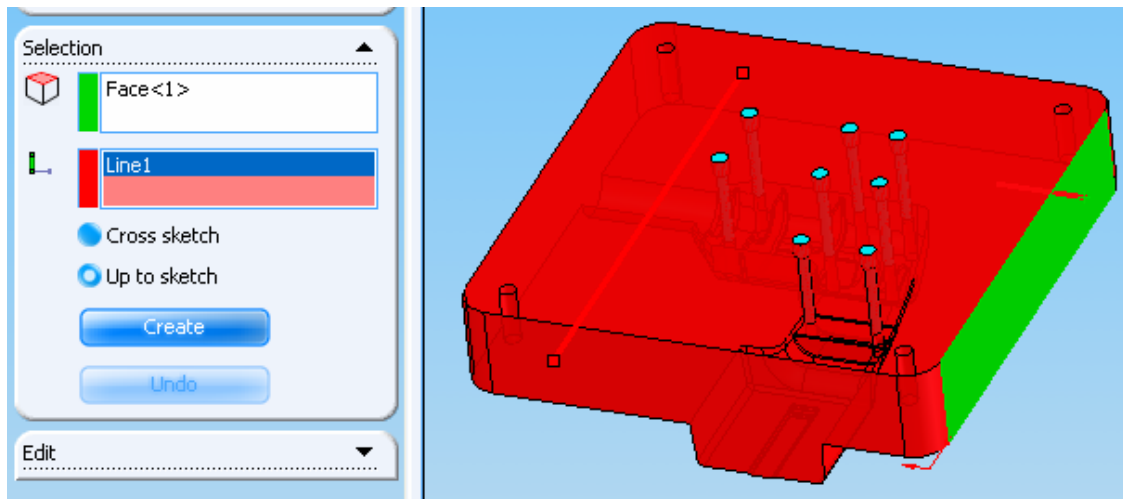


Use Solidworks sketch utilities to put more constrain. For example, edit the cooling sketch, Select the point and line as shown, select Coincident in relation

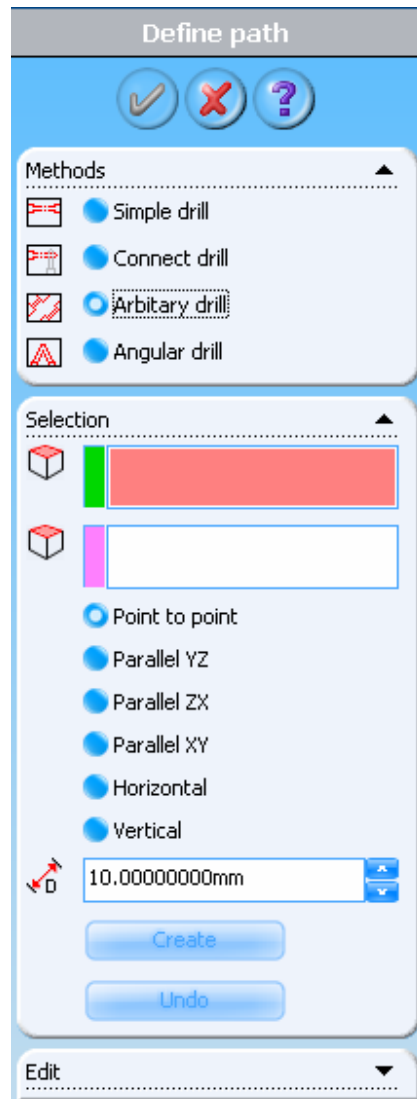


Line segment pass through 2 layers of water channel is obtained.

Up to sketch: Select this option. The newly generated sketch line will end at the connection path.



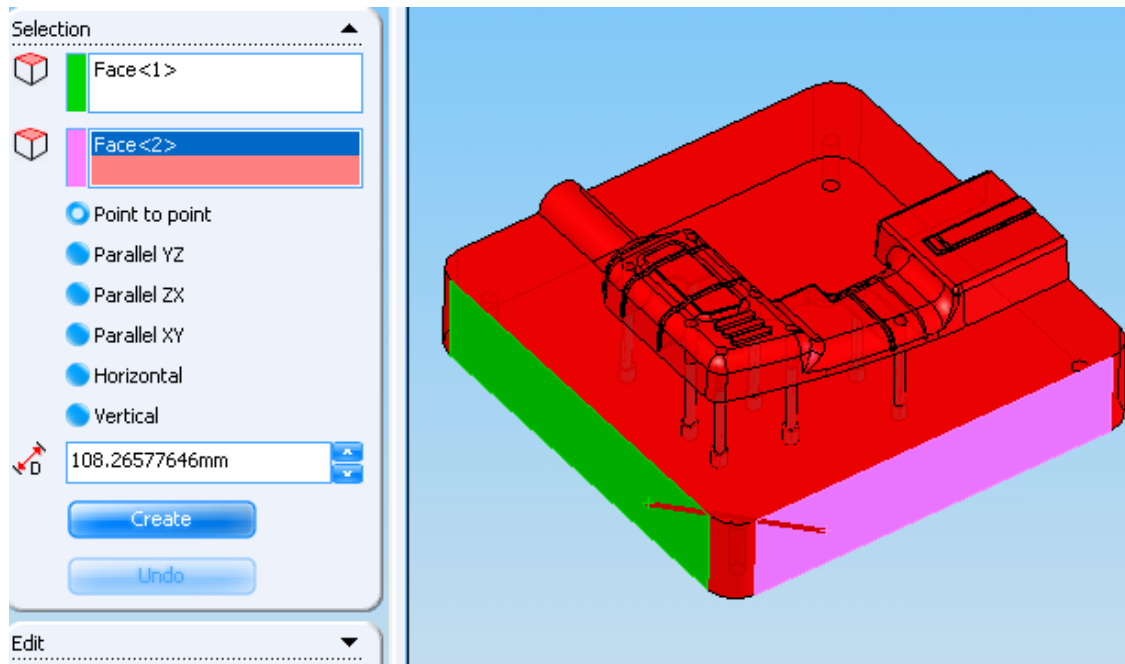
Arbitrary drill: For the case that sketch line is not perpendicular to the selected face.



Drill from: Starting face of the sketch line to be created.

Drill to: Select a face as the ending face of the cooling path, the preview is displayed.

Point to point: Create the sketch line by connecting 2 selection points on the 2 faces.



Parallel YZ: The sketch line (cooling path) is parallel to YZ plane;

Parallel ZX: The sketch line (cooling path) is parallel to ZX plane;

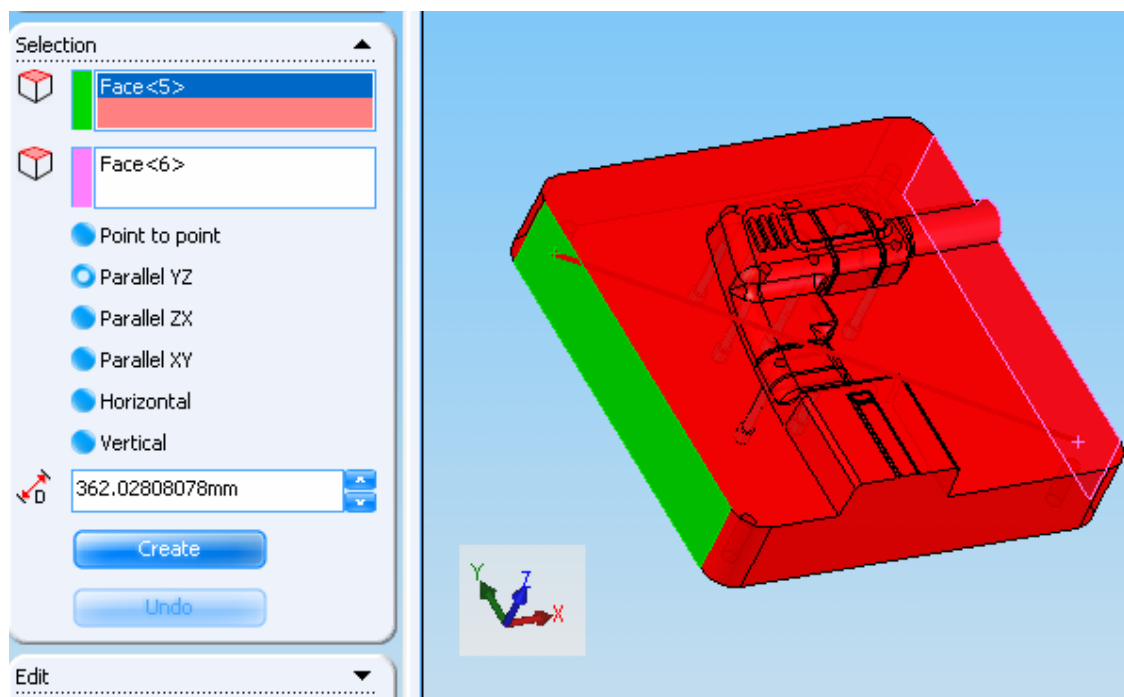
Parallel XY: The sketch line (cooling path) is parallel to XY plane;


Horizontal: the path is parallel to the x-axis

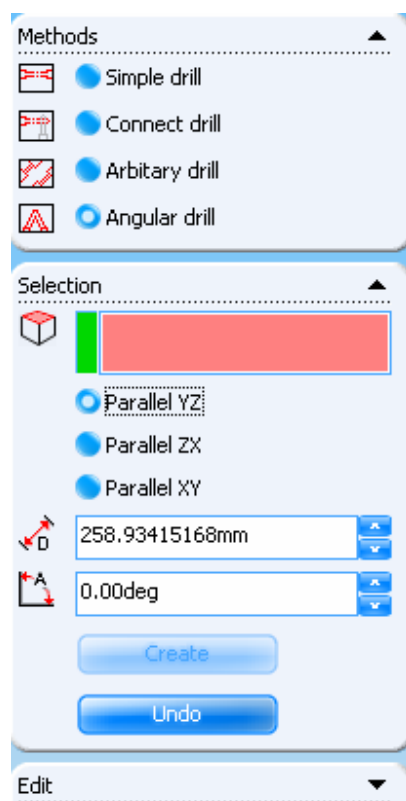
Vertical: created the path is parallel to the y-axis

Drill length: define the length of the sketch line (cooling path) ;

Note: If the selection is not appropriate, there may be contradiction of constrain and the sketch will be over defined.



 Angular drill: V shaped connection:



Drill from: Select a starting face of the sketch line

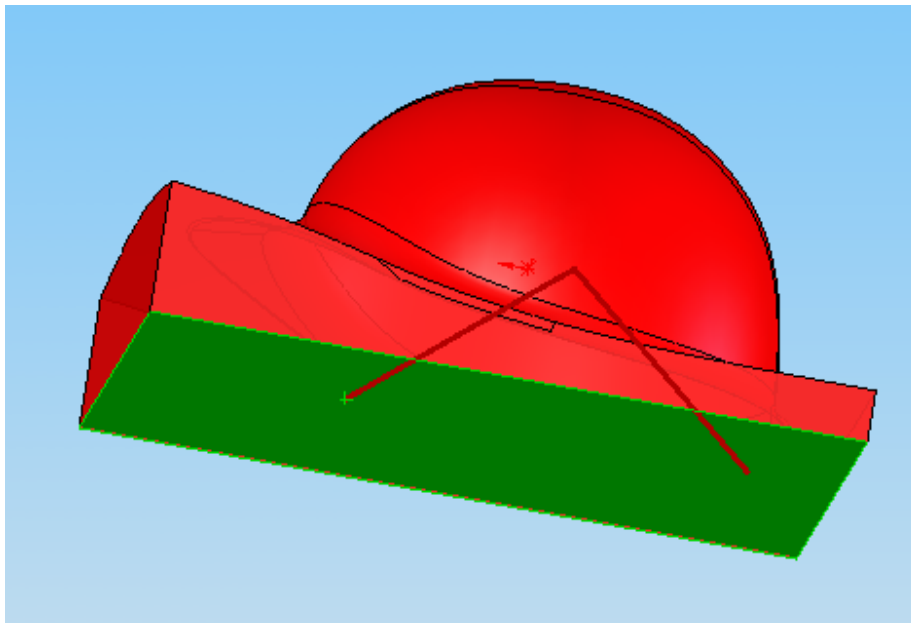
Parallel YZ: the generated cooling path is parallel to the YZ plane

Parallel ZX: the generated cooling path is parallel to the ZX plane

Parallel XY: the generated cooling path is parallel to the XY plane

Drill length: Define the length of the sketch line

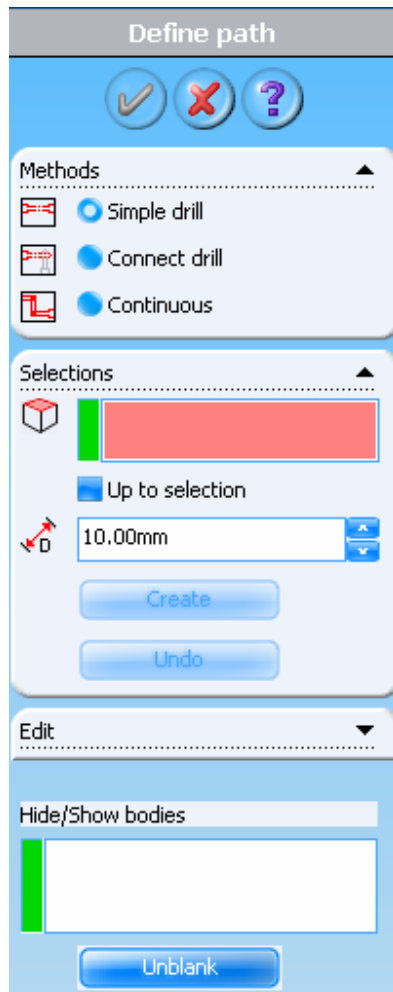
Drill angle: define the angle of the V-shaped connection.



Click **Create** to create the path

After the setting of path is completed, click OK. All the path will be built in a 3D sketch. If the accurate position of the cooling path has to be defined, edit the 3D sketch to become fully-defined.

For assembly file, with **Path** selected, click **Apply** on the cooling manager dialog, the following interface pop outs



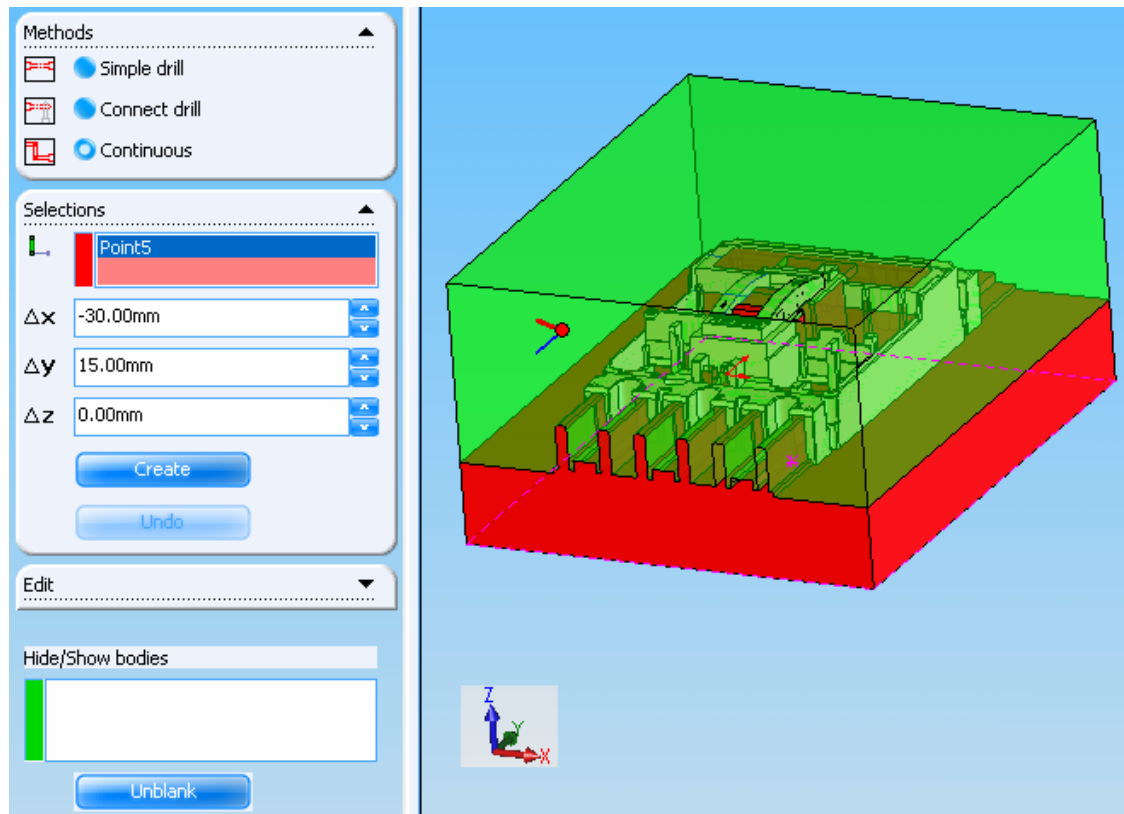
Methods available for creating cooling path on assembly

- Simple drill:
- Connect drill:
- Continuous:

The function of Simple drill and Connect drill for assembly file and part file is nearly the same. The difference is that for assembly file, the Hide/Show function is available hide or show the solid bodies.

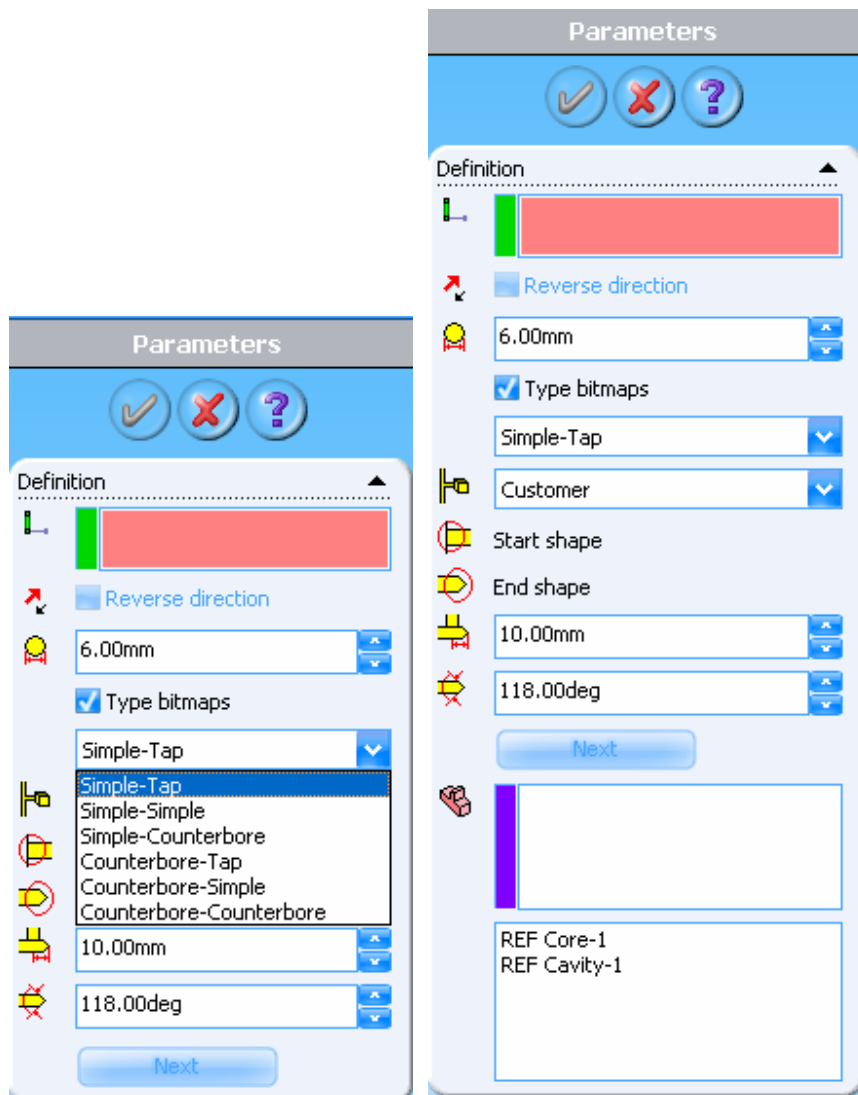
 Continuous: Build the sketch of the path continuously.

As shown below, select the point on a sketch line, input the value of extension in ΔX , ΔY , ΔZ . translation in 2 or 3 directions is accepted. Change the value to its negative to reverse direction. The created line is the line joining the selected point and the translated point.



2. *Parameters*

It is for building the cooling channels. Select Parameters, click Apply to enter the Parameters interface. For part file and assembly file, the interface is slightly different.



When **Type bitmaps** is selected, the types of cooling channel will be displayed



Simple—Tap:


Simple—Simple:


Simple—Counterbore:

Counterbore—Tap:

Counterbore—Simple:


Counterbore—Counterbore:

 : Select the path on the screen, one path can only be selected every time



 : Reverse the direction

 : define the Diameter of the water channel

Type biymaps: Display type of cooling channel in graphic area.

 : Select the configuration of the channel diameter. Select a type, other parameters will change corresponding to the type. User can check the data of different type of cooling channel in the installation directory res \ Cooling Parameters.xls, configuration can be added and changed also.

	A	B	C	D	E	F	G	H
1		Diameter	Start_diameter	Start_depth	End_diameter	End_depth	Extension	Angle
2	Customer	6	12	15	12	15	10	118
3	6mm	6	12	15	12	15	10	118
4	10mm	10	20	15	20	15	15	118
5	12mm	12	25	15	25	15	20	118


Strat shape: type of starting shape of cooling channel, which includes Simple  and Counterbore .


 : Counterbore diameter

 : Counterbore depth

End shape: type of ending shape of cooling channel, which includes Tap , Simple , Counterbore .


When Tap is selected,

 : Over drill length

 : Angle at bottom

When Counterbore is selected

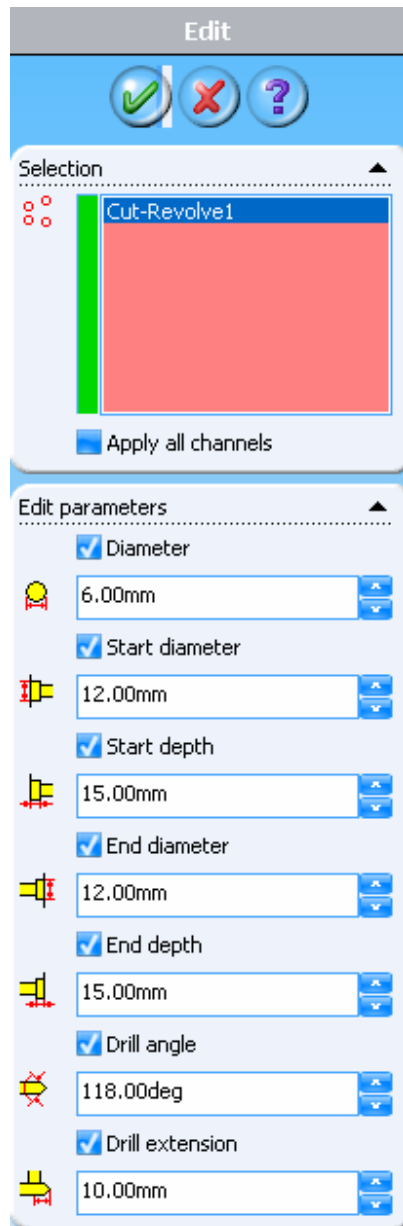
 : Counterbore diameter

 : Counterbore depth

After setting a segment, click **Next** to proceed to set the other segment.

3. Edit

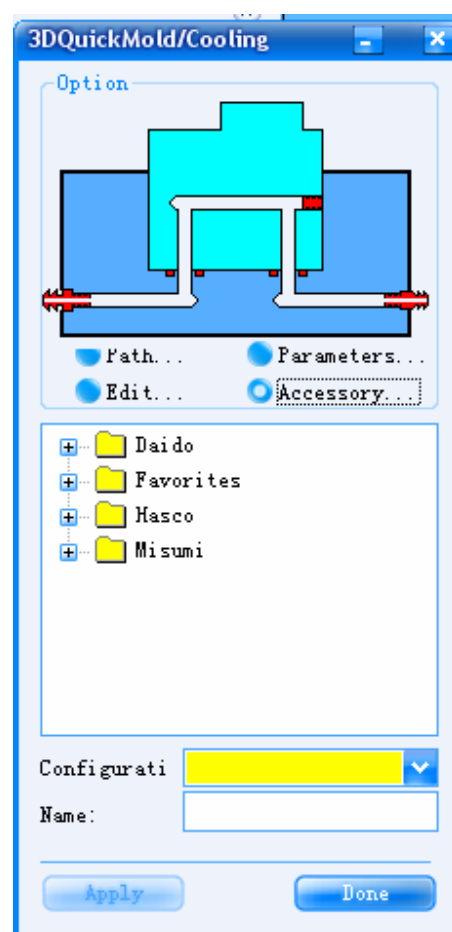
After setting the Parameter, user can enter Edit to edit the Parameter. The dimension of the cooling channel can be edited here. The position and the relation can be edited using the sketch function of Solidworks.

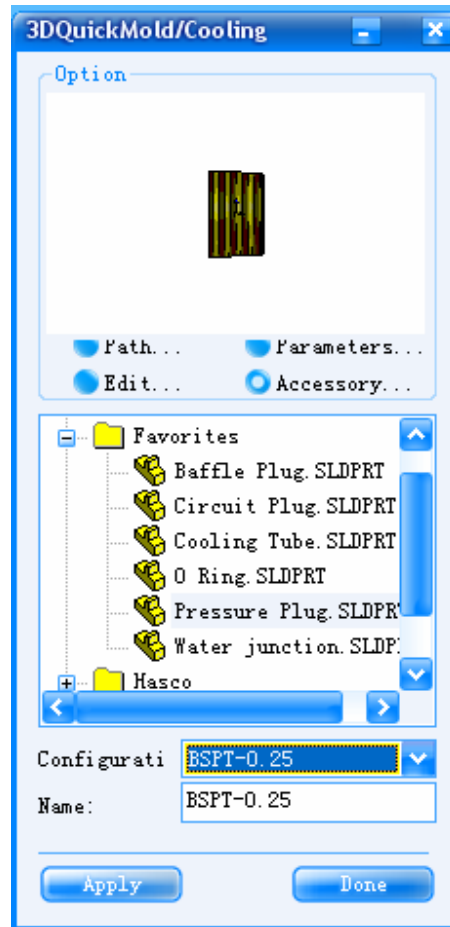


Apply all channels: Apply the changes to all the cooling channels.

4. Accessory

Some typical standard parts for cooling such as End plug and O-ring are provided. The adding of these parts is required to perform in assembly files.





Preselection is required for adding the cooling accessories. Except Circuit Plug, adding other parts requires preselecting a circular edge. Click Apply to add.

Multiple circular edges can be preselected to add the cooling accessories.

Adding a Circuit Plug is required to preselect a circular edge and a sketch point.

Select the required accessory to add, the preview of the part is shown in the dialog.

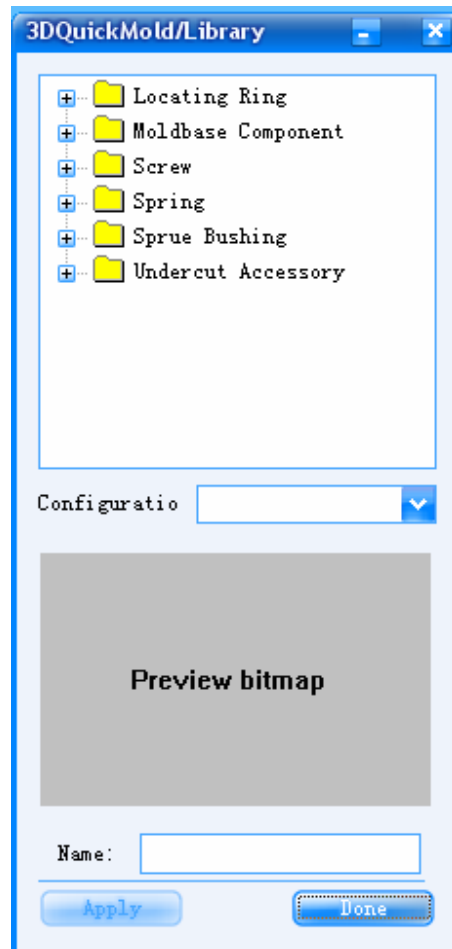
Select type in the configuration field.

Input the name of the part in the name field.

9. Library Manager

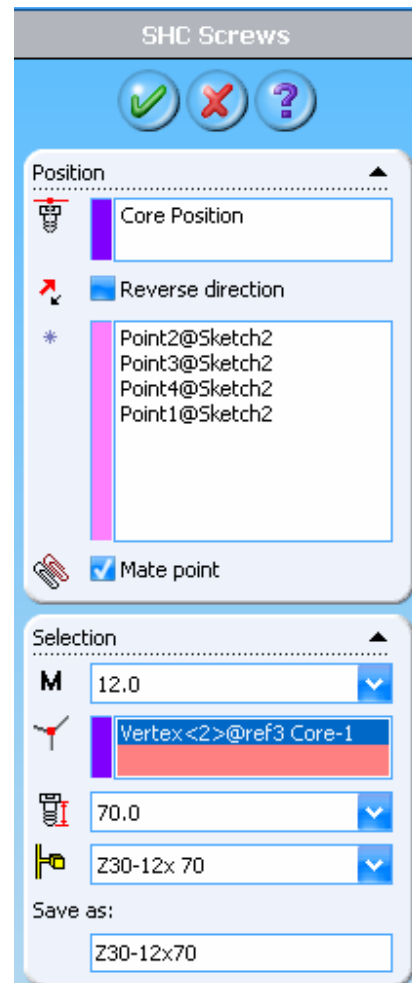
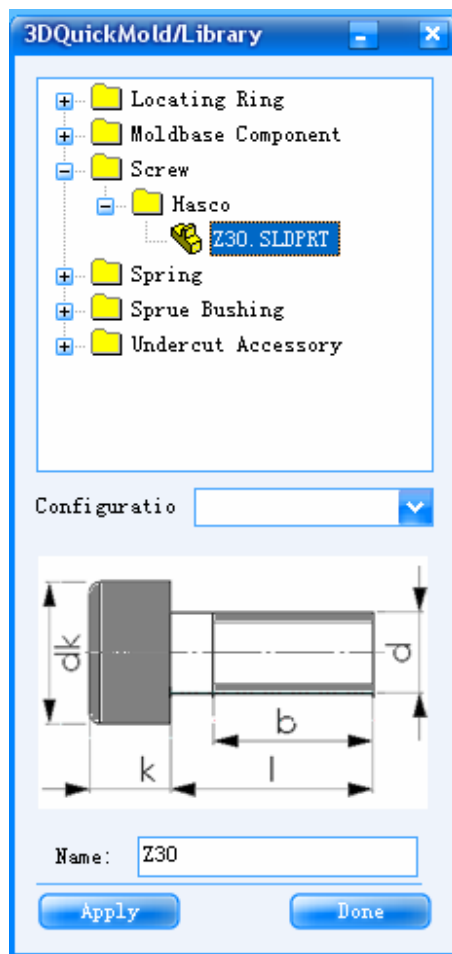
To increase the design efficiency and reduce the duplicating procedure, 3DQuickMold provides a standard library manager. Those libraries include locating ring, sprue bushing, screw, spring and parts for slide and lifter.


Adding standard parts from the Library to mold assembly can only be used in Assembly file.




1. Adding Screw


To add a screw, select a type, a part bitmap is shown. Click **Apply**, the property manager of the screw pops out, the size, position, plane and relation can be defined.




 : Screw mate reference, please select a planar face


 : Reverse the screw direction

 : the position of the screw, this can be a sketch point or a vertex of a line.

 : mate point: When this option is checked, a concentric condition is applied to the the screw and the selected point

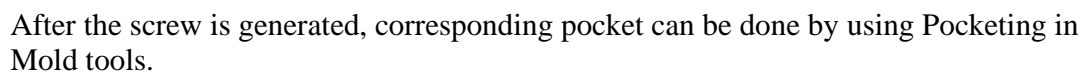
 : The screw diameter

 : length reference: A reference vertex to determine the screw length.

 : length of the screw

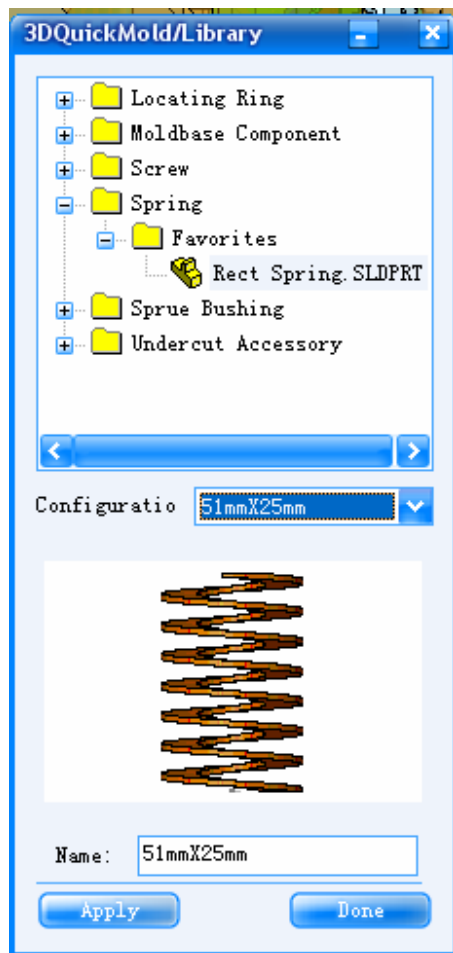
 : Screw configuration : can be obtained by selecting the length and diameter.

Save as: Screw Name





To add a spring, at least, two mate constrains should be applied.

132




Position


: spring mate reference. It could be the spring origin or the XY plane (Front) in the spring model.

: target mate. Mate reference on the assembly model. If the above selection is the spring origin, this face should be a cylindrical face. If the above selection is a plane, accordingly, a planar face is required to be picked up here.

Undo last mate: Click Undo last mate to cancel the mating relation.


Size

: spring configuration

: length of spring

: Thickness of spring

: inner diameter of the spring

: outer diameter of the spring

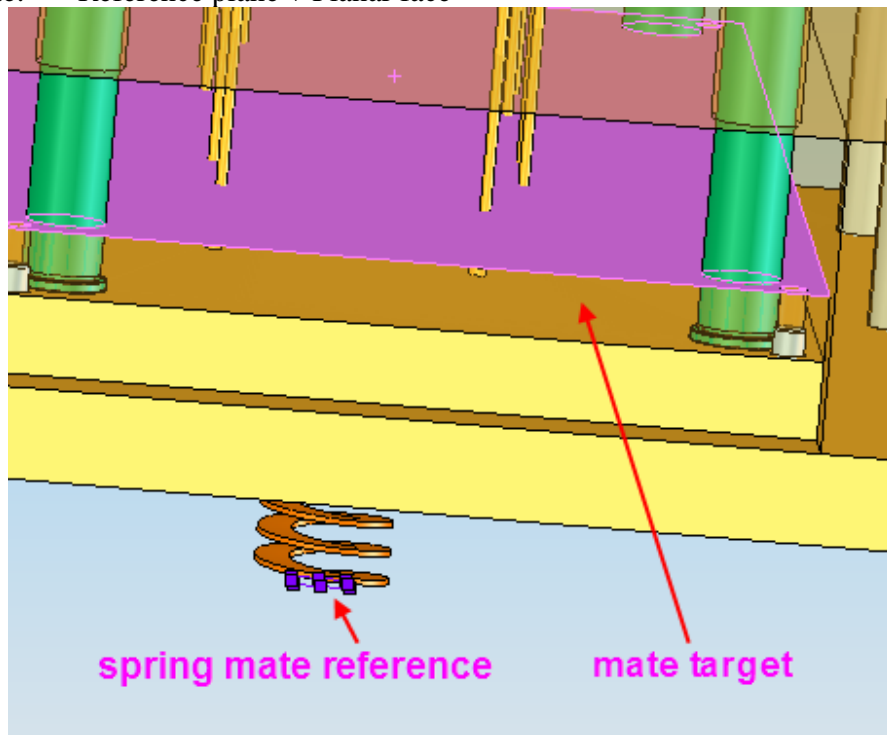
Naming

: Prefix

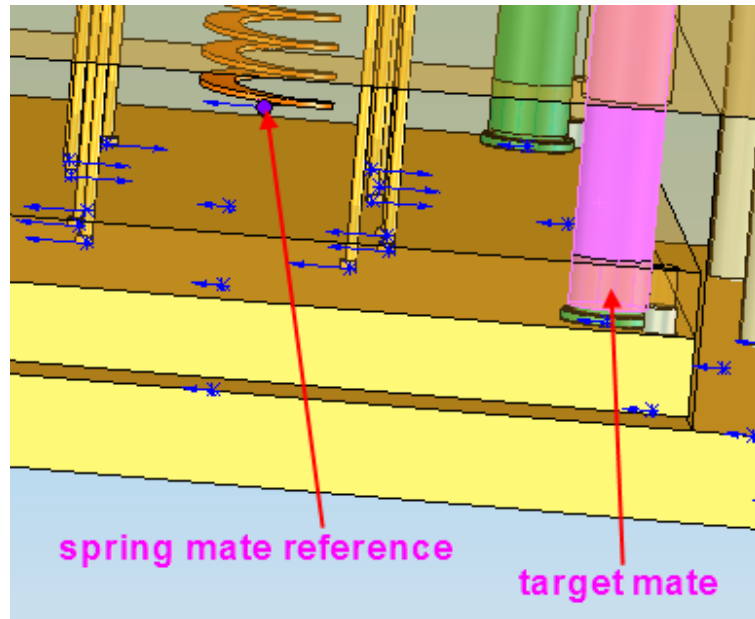
: Suffix

Save as: spring name

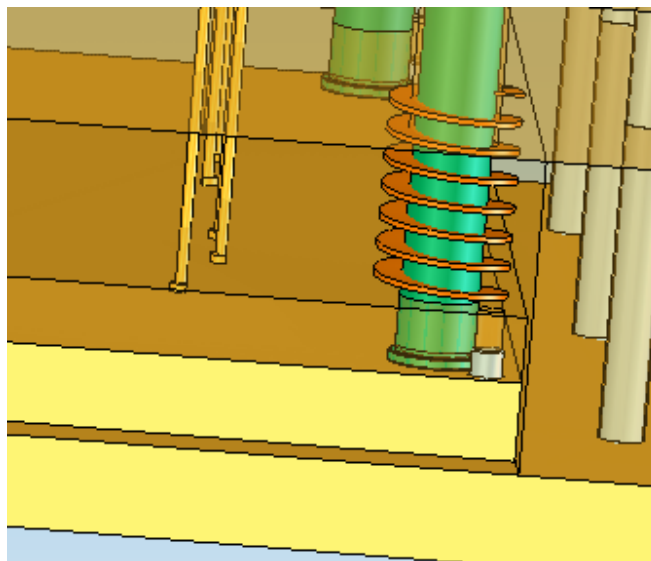
First mate: Reference plane + Planar face



Second mate: Spring origin + Cylindrical face

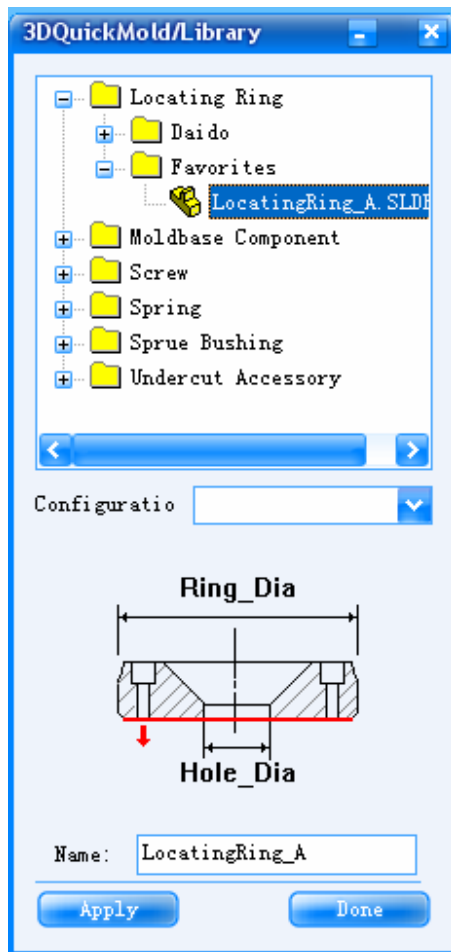



Define the dimension and click OK




After the spring is added, pocket for spring can be done by using Pocketing in Mold tools.


3. Adding Locating ring




 : the face on top plate that mate with the locating ring

 : the face on the locating ring that mate with the top plate

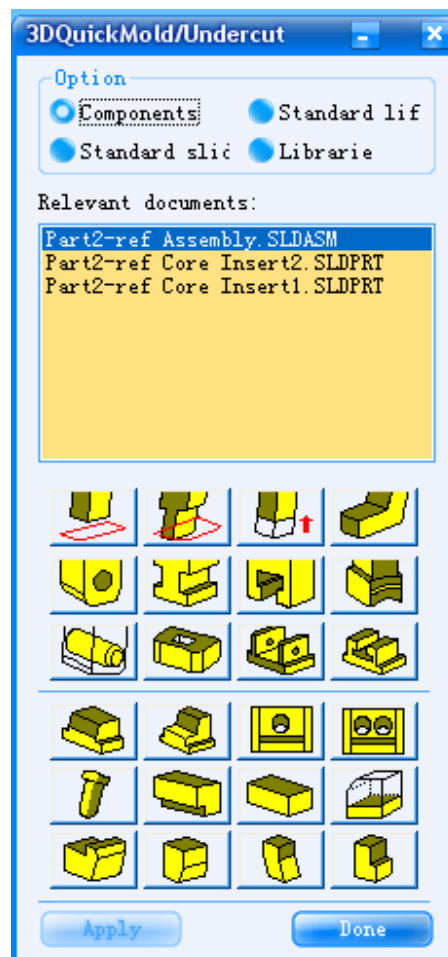
 : A cylindrical face of the locating ring

 : the depth that the part of locating ring offset from the top plate

 : Configuration of locating ring

10. Undercut Manager

To design the side core mechanism including the slide and lifter design, customer can chose the 3DQuickMold build in slide and lifter design, user can also have their own design and build up their own standard for the undercut mold release mechanism.



There are two different methods to design slide and lifter

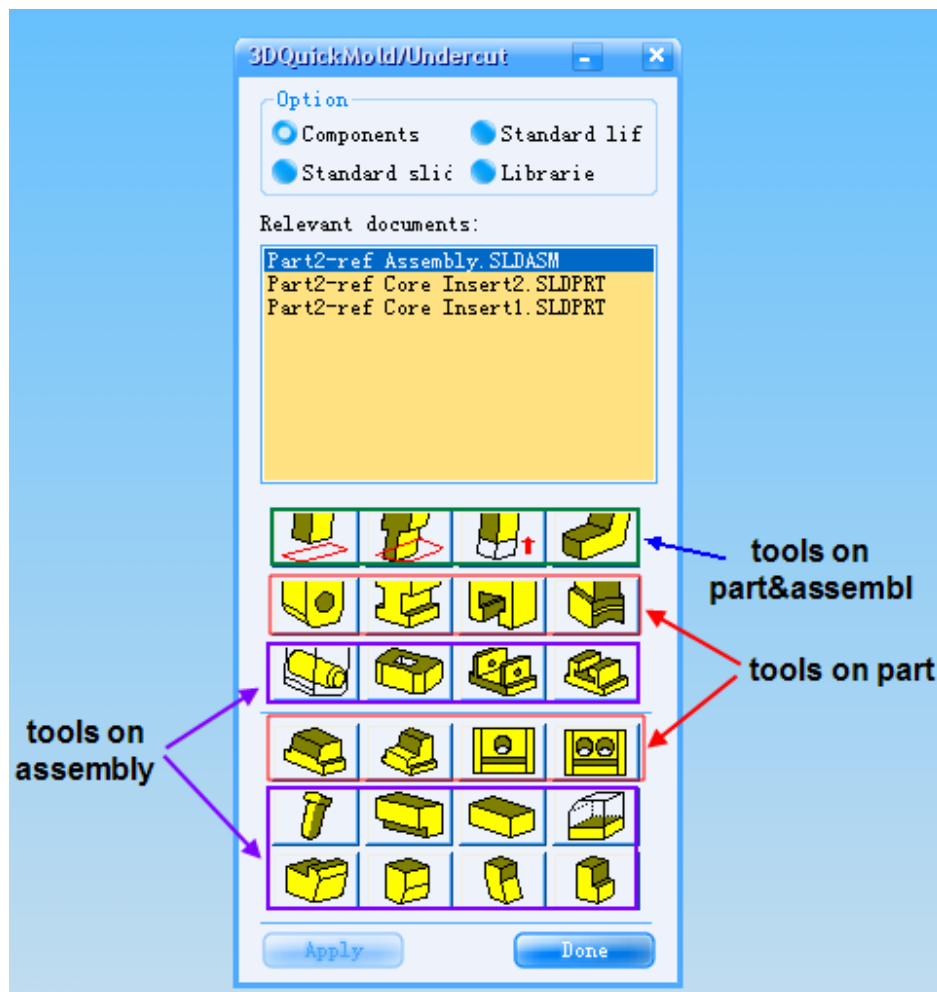
One is to use the whole set of standard parts, where the angular pin, guide rail and slide body are all imported to the assembly. Standard lifter and Standard slide are of this method, this approach could be seen in many other mold design solutions, it looks quite effective, but less flexibility.

Another method is to add parts one by one which enhances the flexibility. It is implemented when Component option is selected.

These two methods both have their own advantages, users can choose between them according to the need.

1. Components

Some icons below are used to add special features for slide and lifter body. Others are used to add some common slide and lifter components to the assembly.



Relevant documents: Quickly activate the working document by double clicking the file name in the list.

There are 6 rows of icons totally as the above picture shows.

The first 3 rows is used for lifter design, the last 3 rows are used for sidecore design.

Icons on the first row are used on part and assembly files, icons on the second and the forth row are used on part files, icons on the third and the fifth row are used on assembly file.

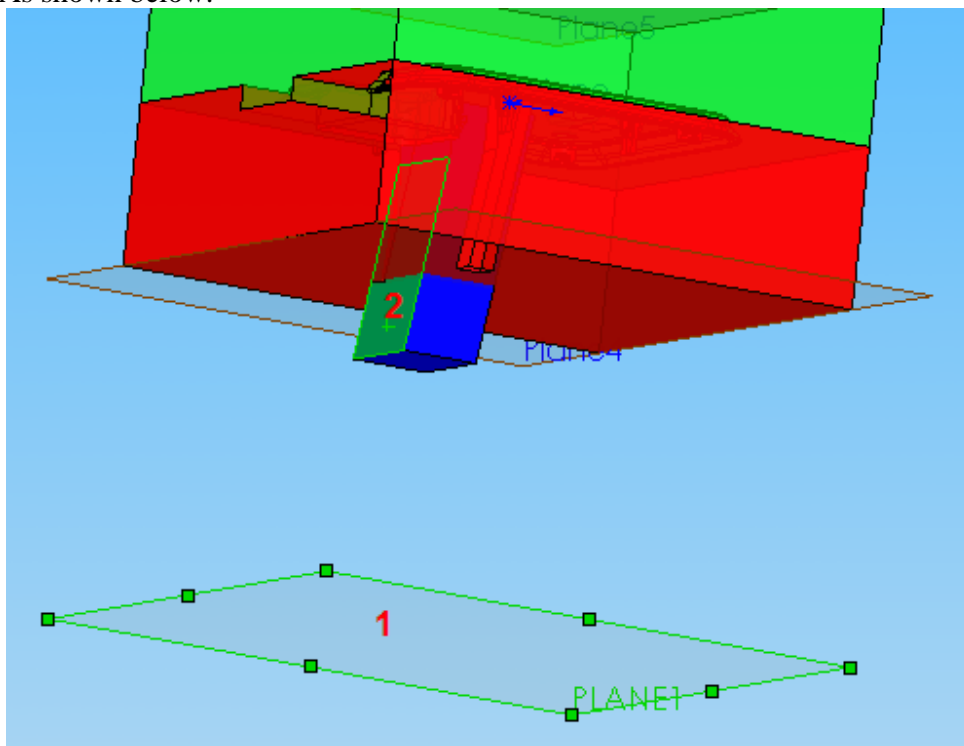
Lifter design tools icon:

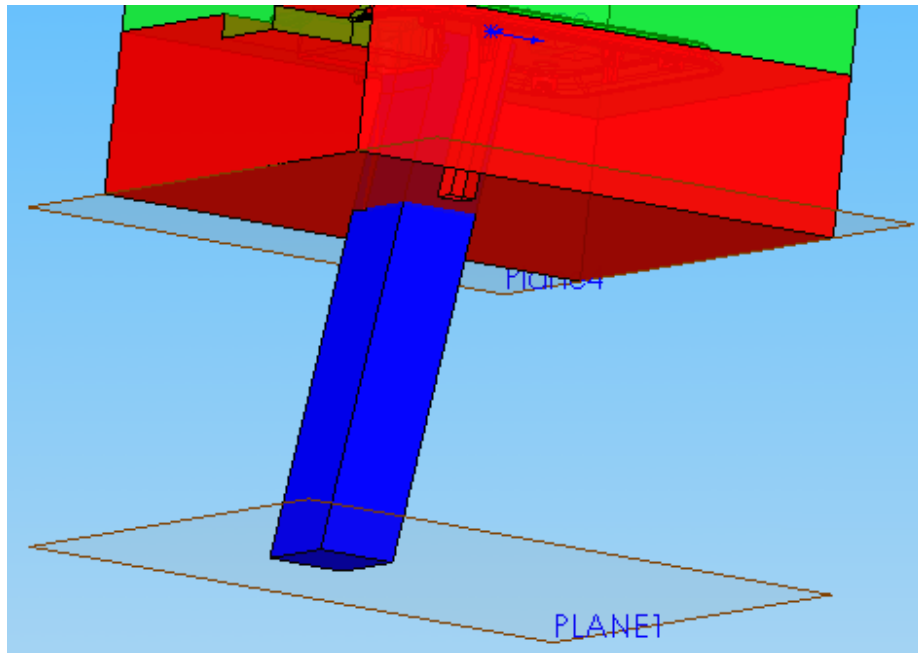
The following 4 icons can be used on the assembly or part.



: Extend lifter body to the selected plane

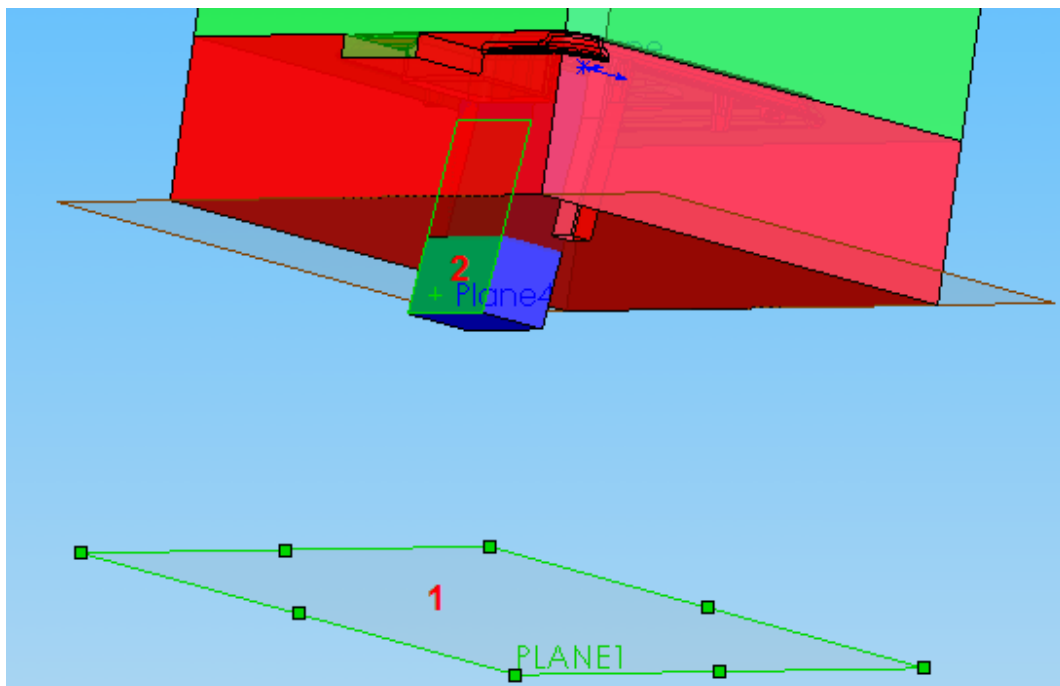
Method: select a reference plane and the inclined face on lifter body in order, click the icon. As shown below.

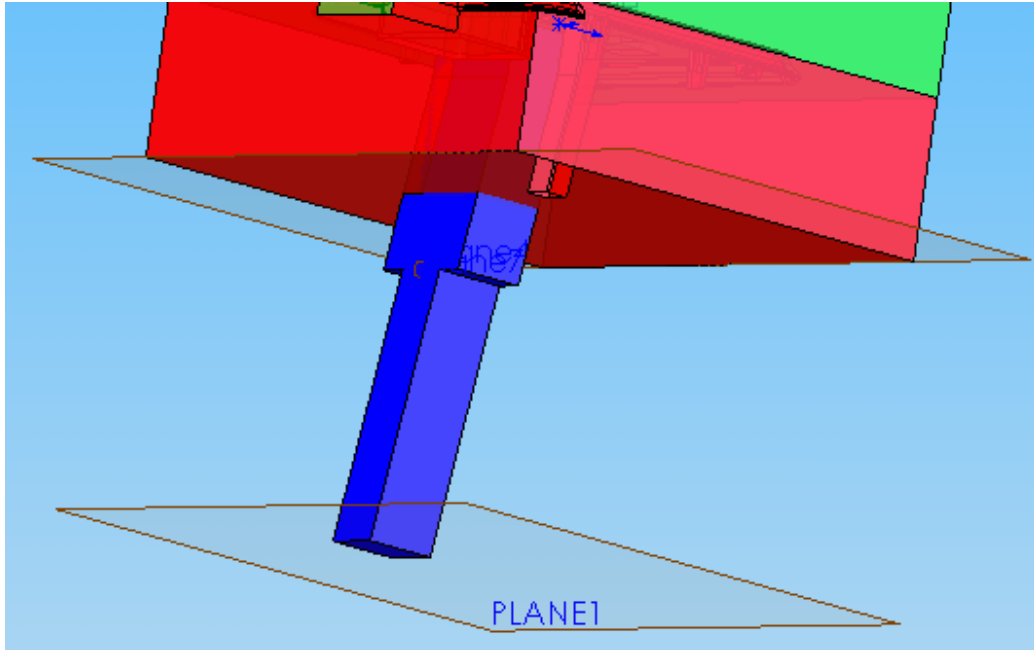




: Extend lifter body to the selected plane but using a narrower profile.

Method: Select a plane and an inclined face on the lifter body in order, click the icon. As shown below.

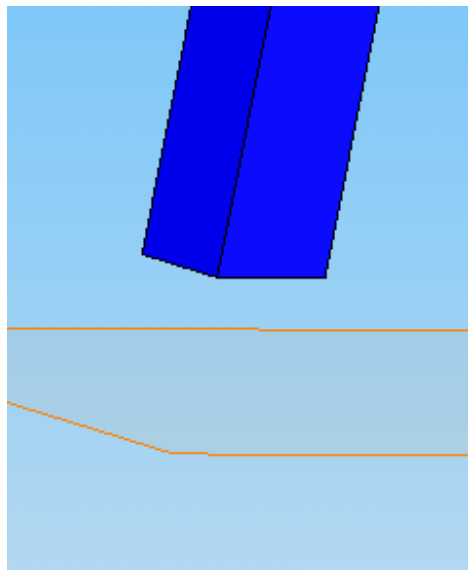
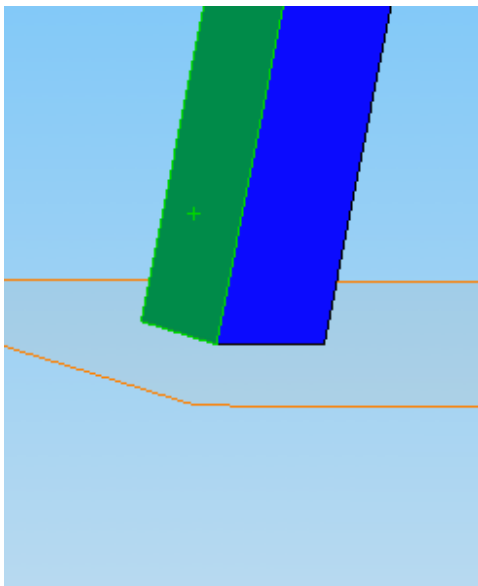




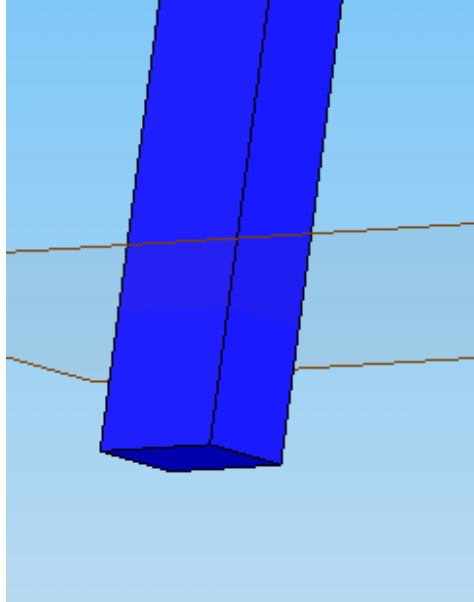
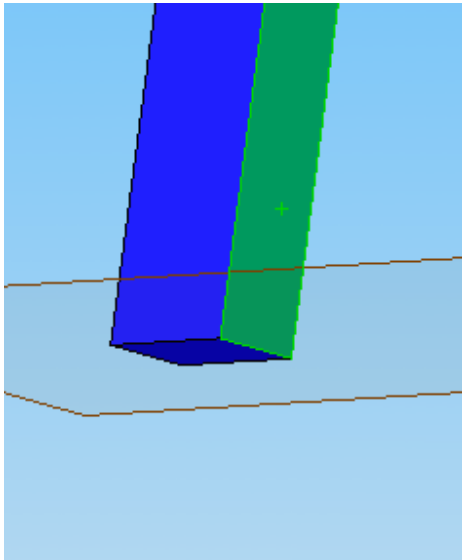
: Extend or shorten the length of lifter body.

Different result depends on the selection point.

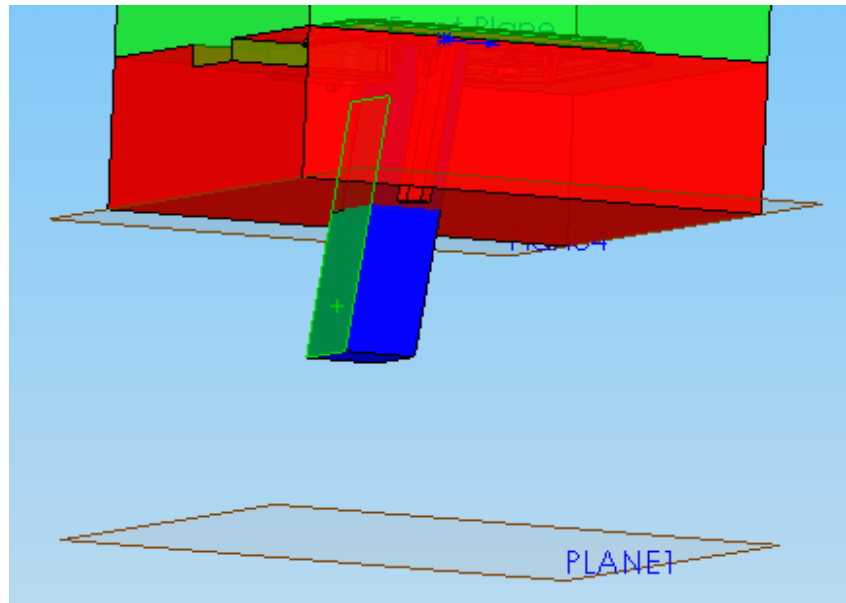
1. To shorten the length, Select the inclined face that is at acute angle with the lifter bottom face, click the icon. As shown below,

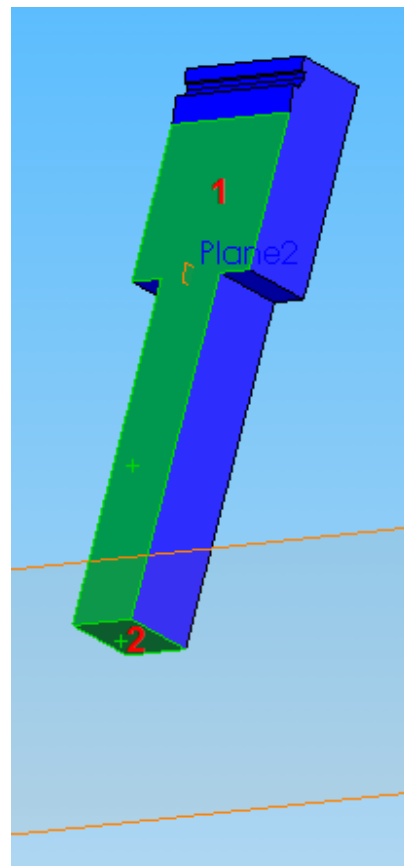
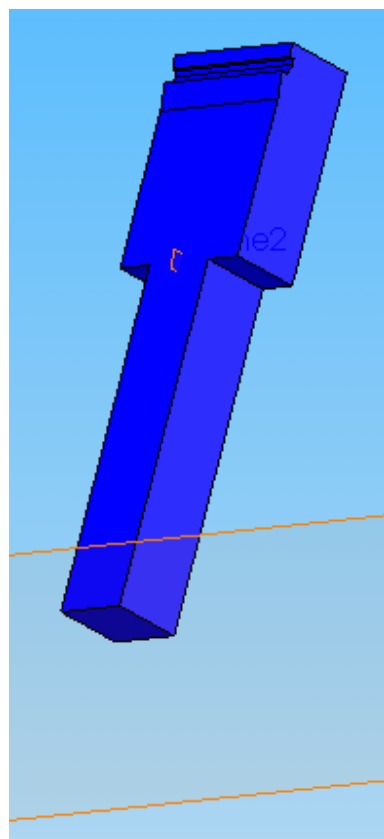
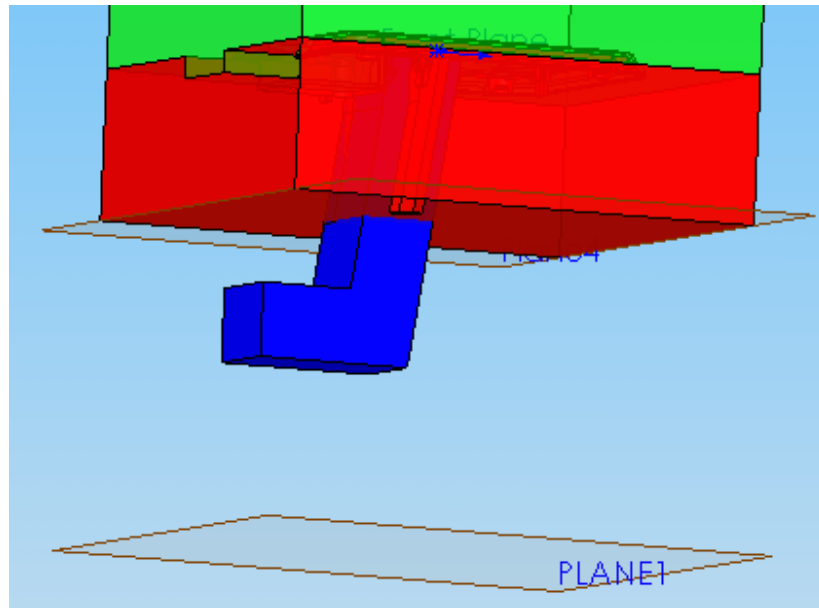




2. To extend the length, select the inclined face that is at obtuse angle with the lifter bottom base, click the icon. As shown below,

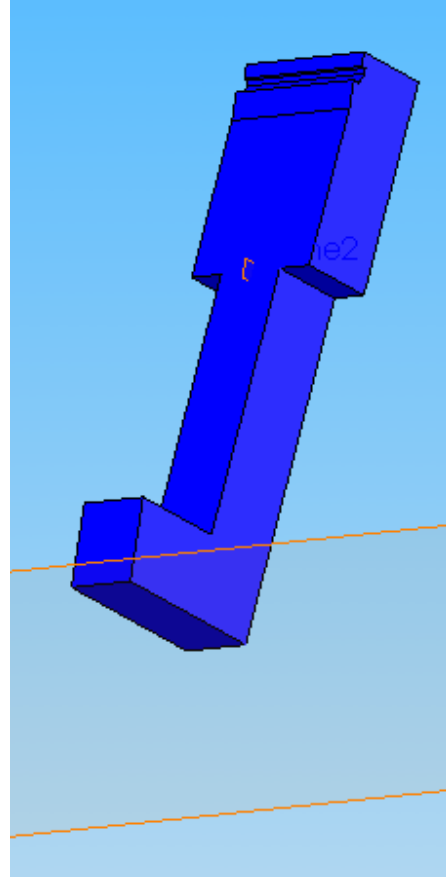
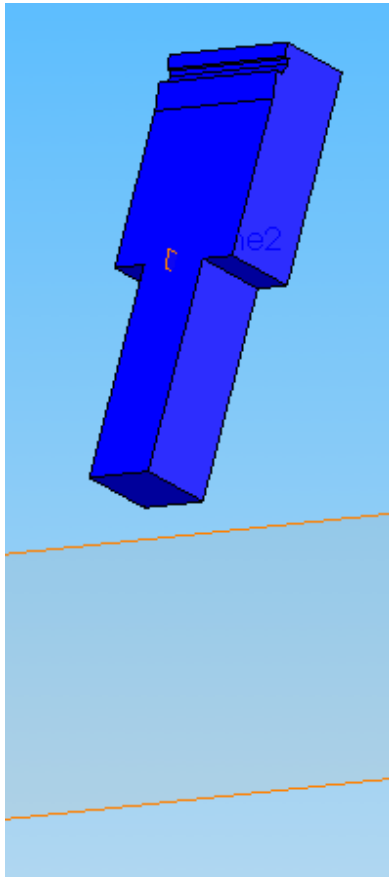


: Used to create a feature on a special lifter body as the picture shows. To do this, select the inclined face that is at acute angle with the lifter bottom face, click the icon. As shown below,





Click  and  separately, the result are shown as follow:

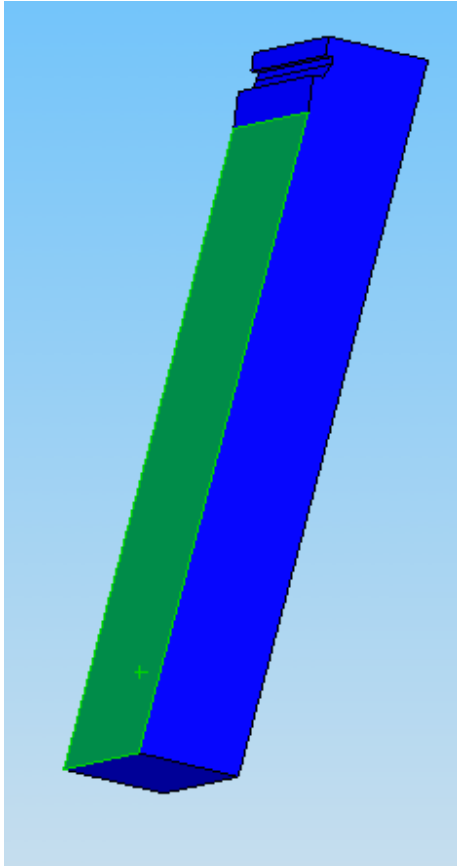


The following four icons are applicable on part file only, they are used to modify the lifter bottom area to suit different lifter assembly structure.

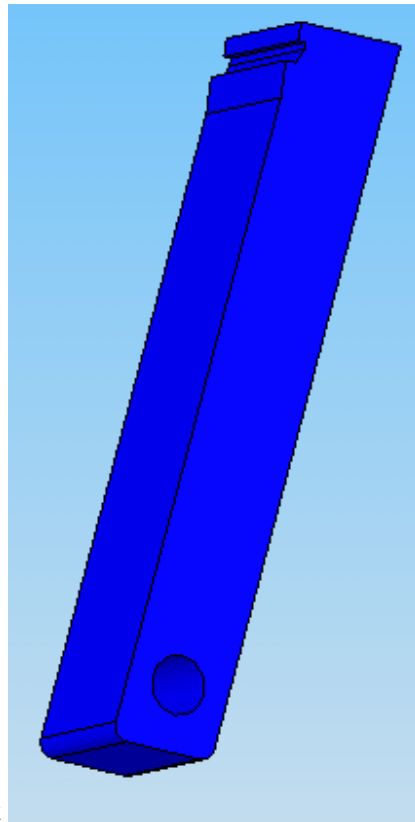


: Type 1

Method: select the inclined face, click the icon.

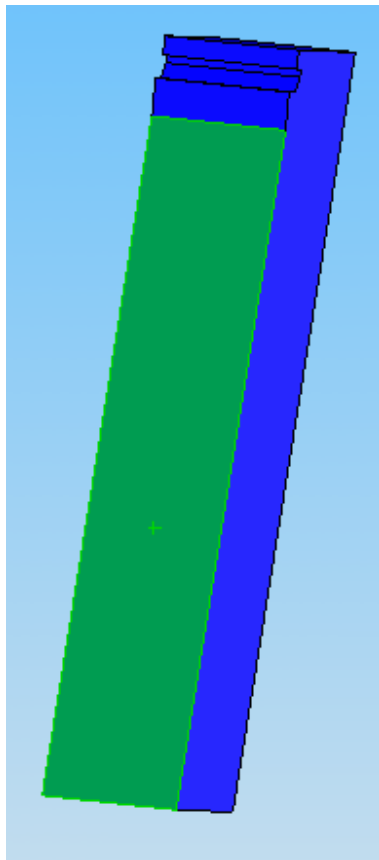


Result

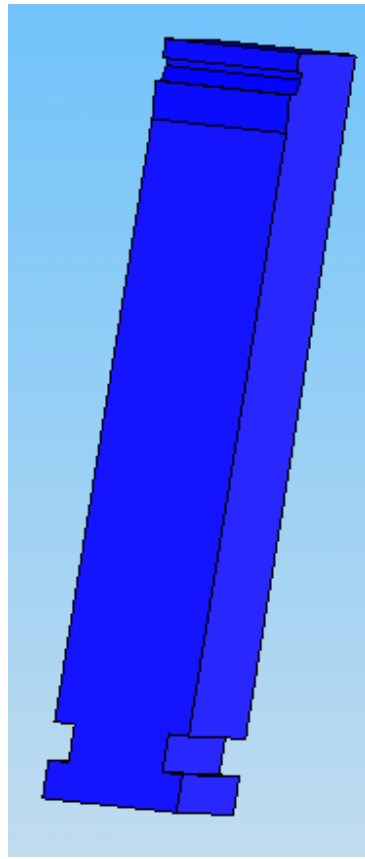


: Type 2

Method: select the inclined face, click the icon. As shown below.

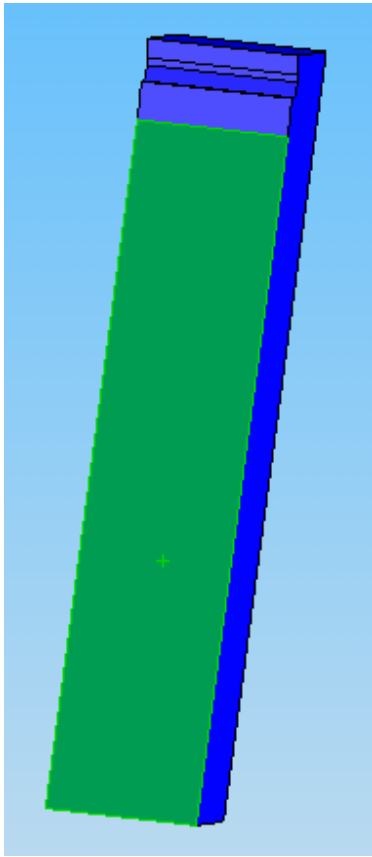


result

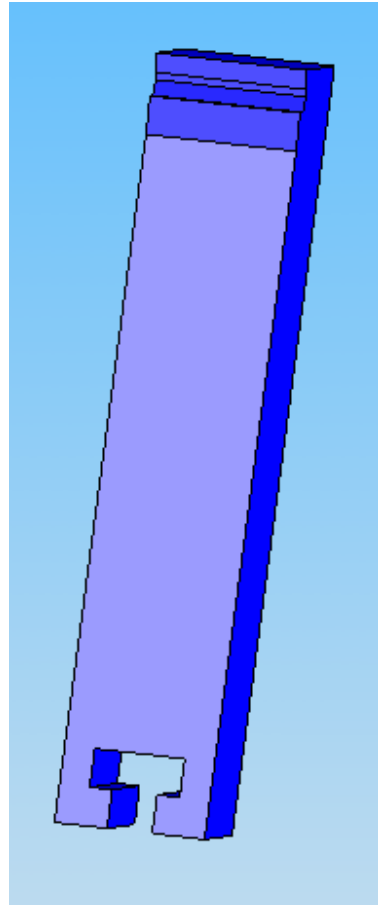


: Type 3

Method: select the inclined surface, click the icon. As shown below.

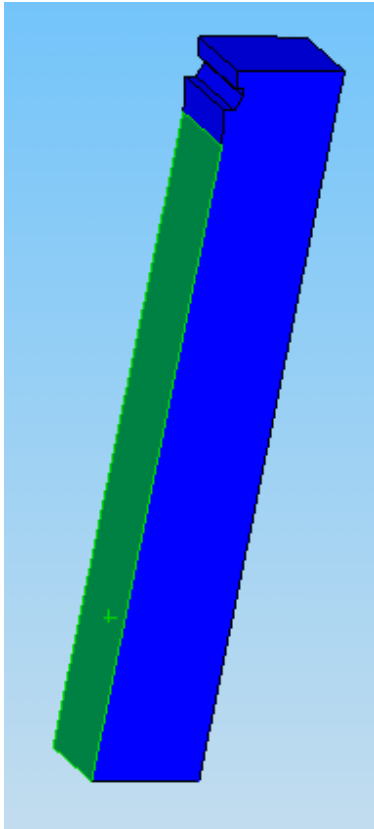


result

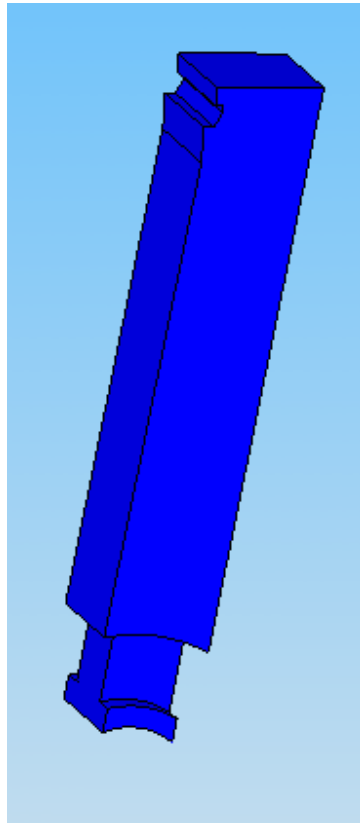


: Type 4

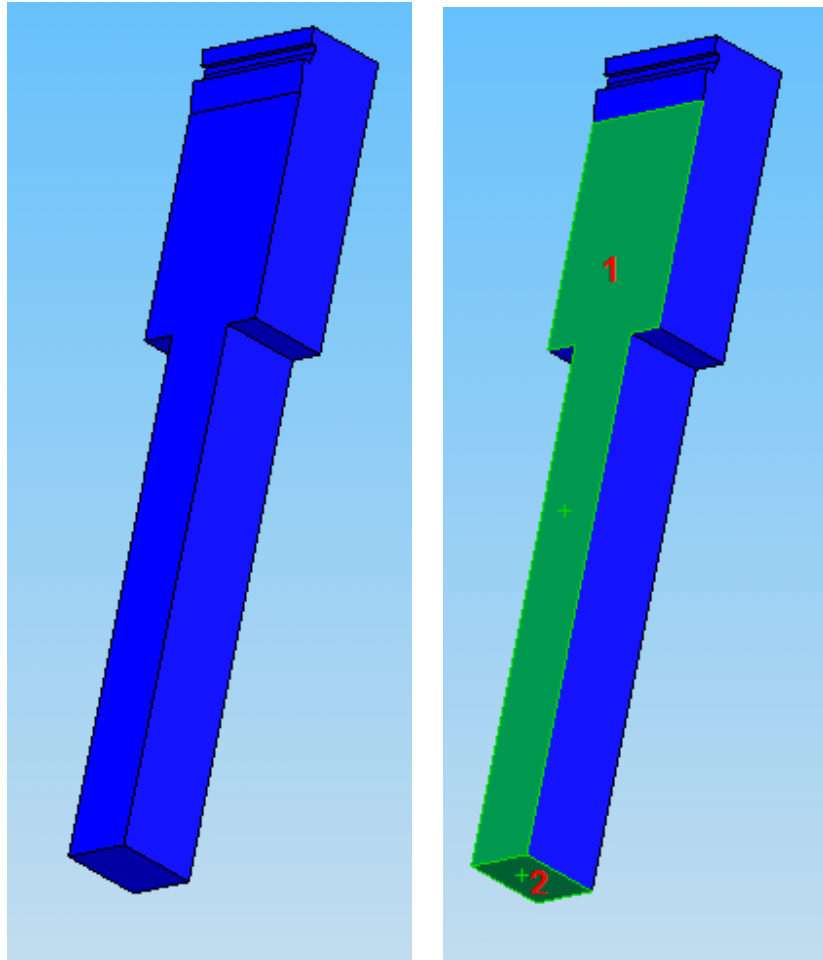
Method: select the inclined surface, click the icon. As shown below.



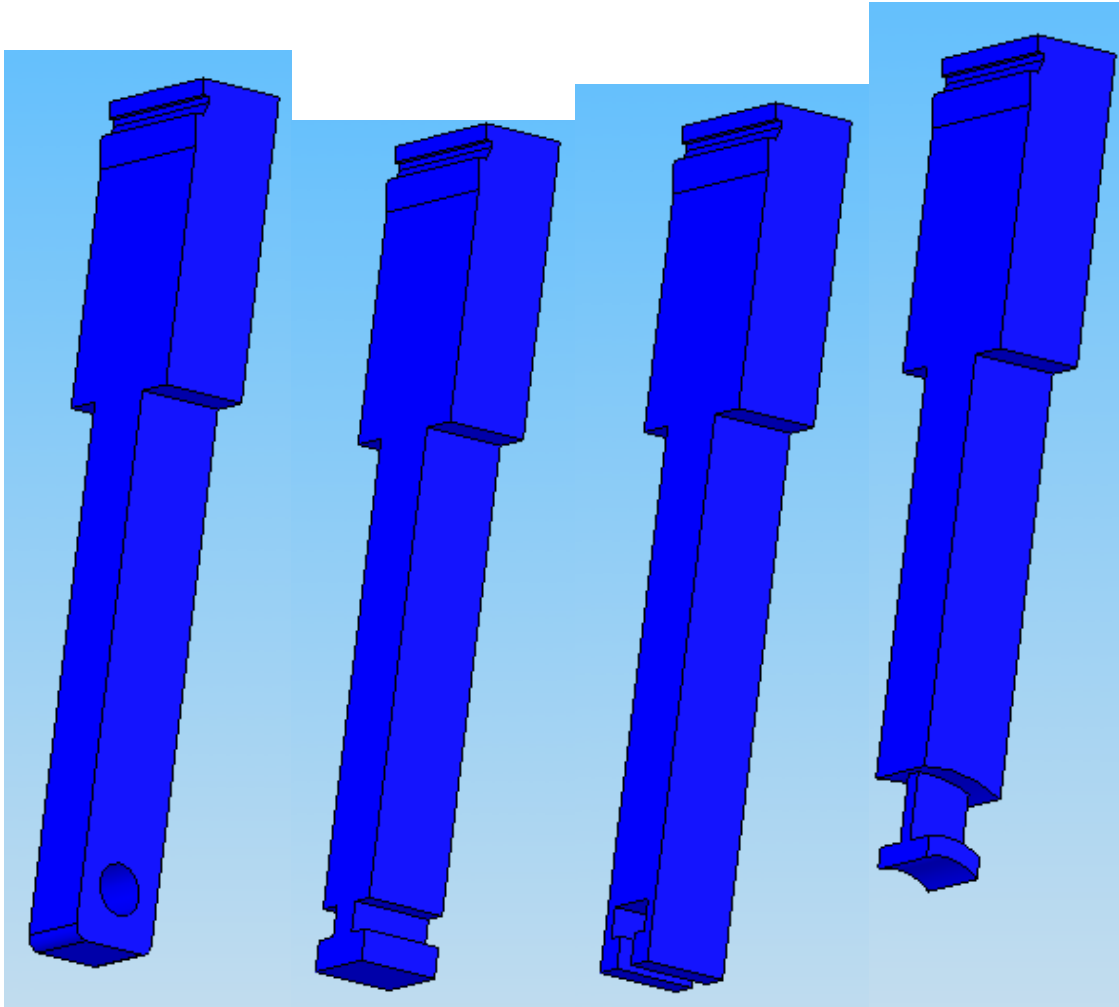
result



Note: for the following type, select an inclined face and a bottom face in order, click the icon to finish.



After selecting two faces, click each icon respectively and the result are shown below.



The following four icons are used on assembly file, using those functions will add some extra components to current assembly



Pin



Lifter house



Lifter house

Slider design functions icons:

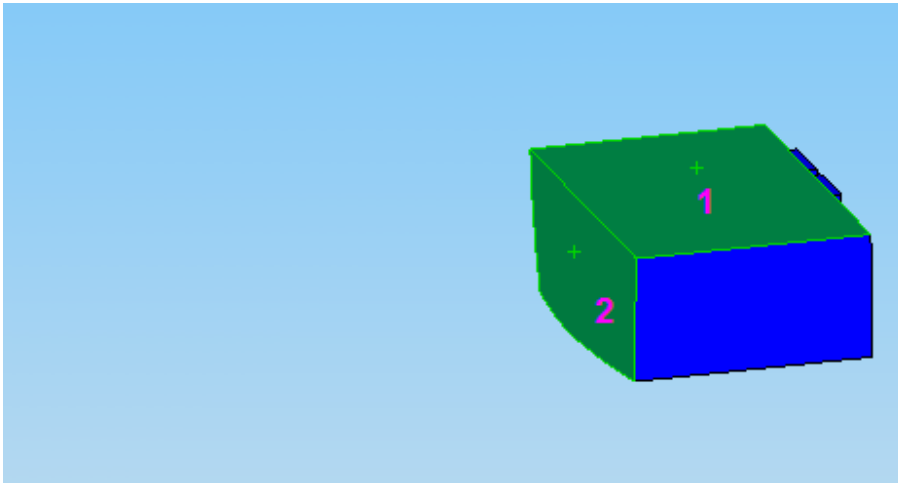
Icon for parts:



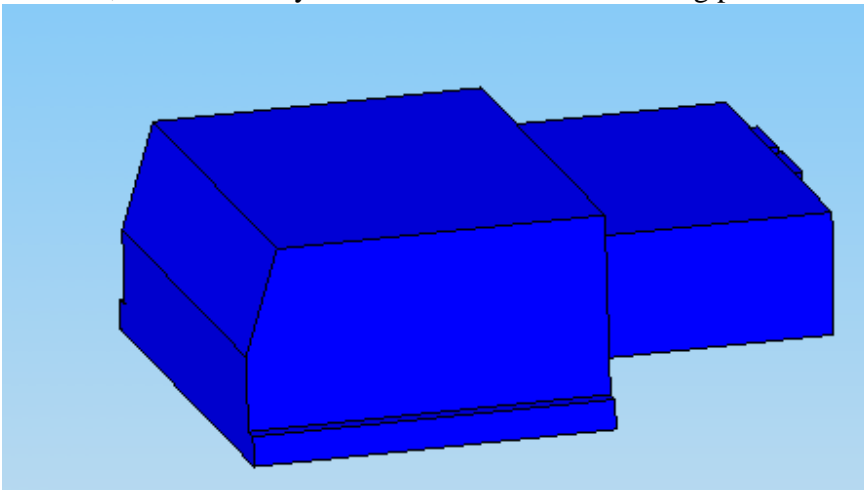
:Slider body type 1

Method: select two planes, click the icon, the first plane determines the orientation of slider body, the second one is the face that the slider body will attach to. As shown below.

Selections as follows in order

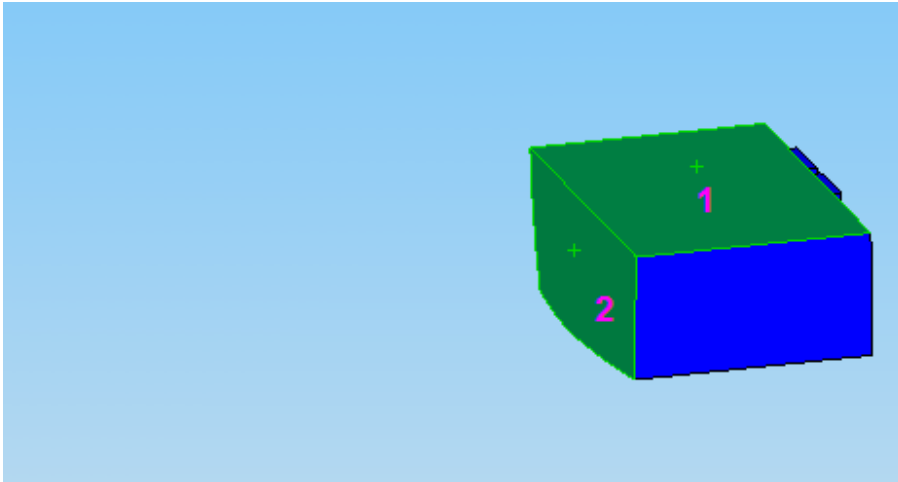


Click the icon, the slider body will be created as the following picture shows.

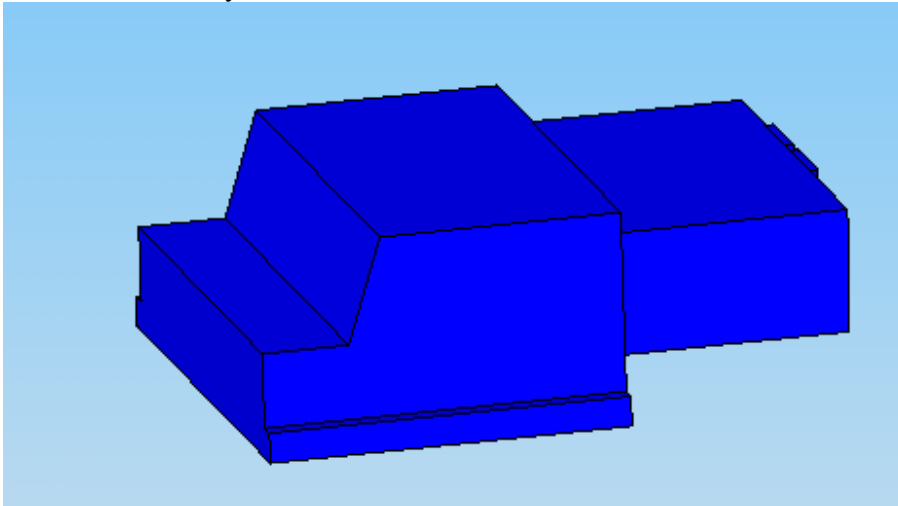


:Slider body type 2

Method: same as type 1

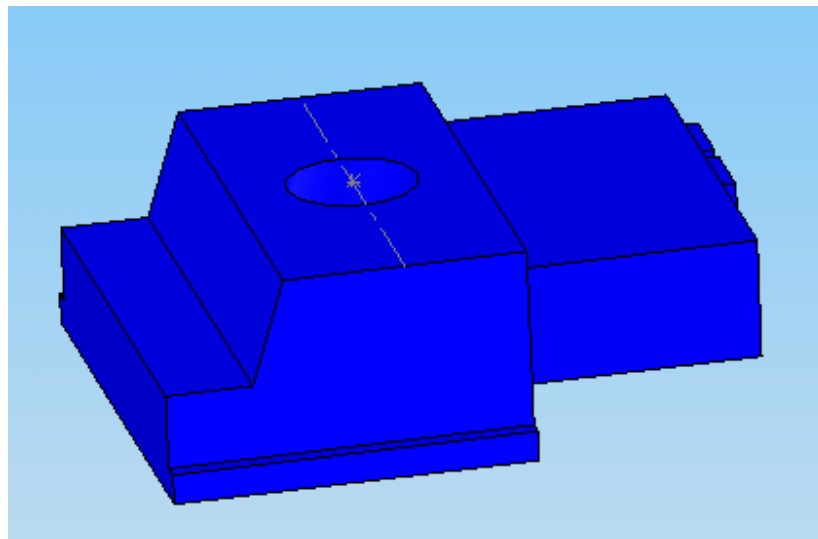
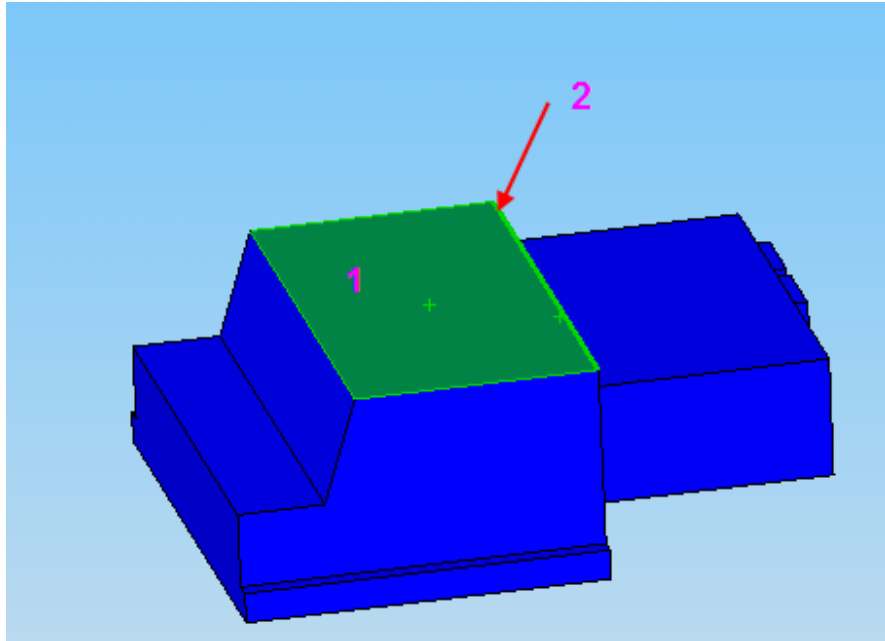


A different slider body will be created

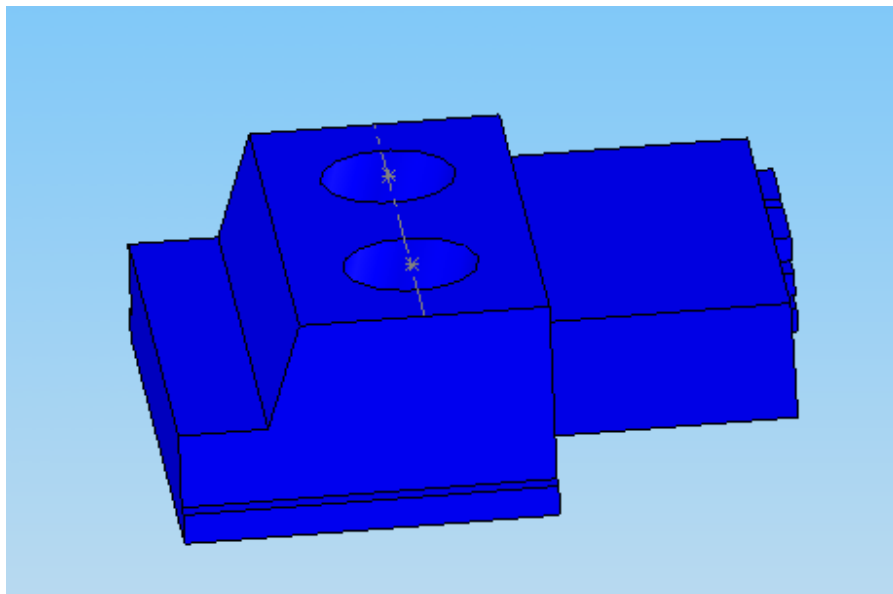
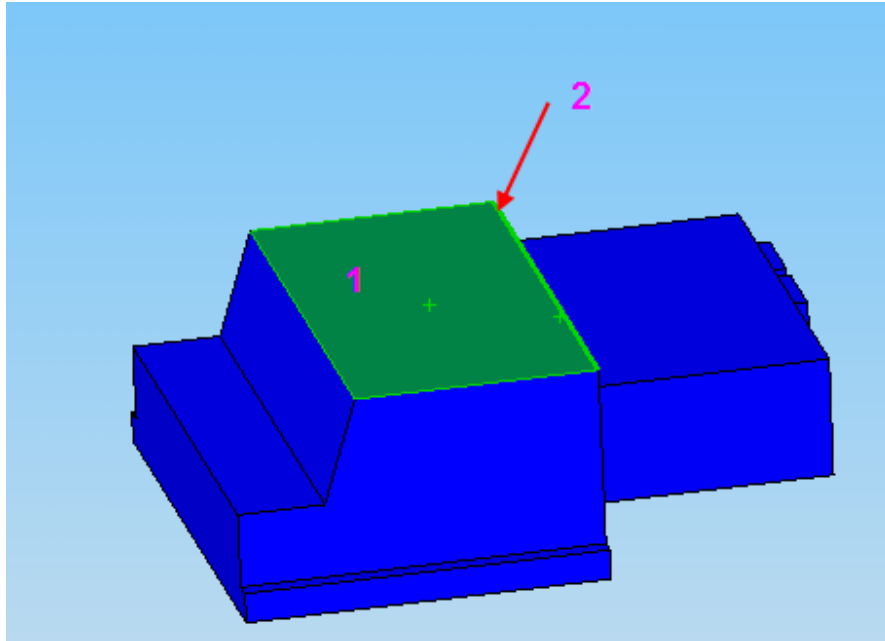


: Create a hole on slider body for angular pin.

Method: select a face and an edge in sequence as follows, click the icon.



: Create two holes on slider body for angular pins.
Method: select a face and an edge in sequence as follows, click the icon.

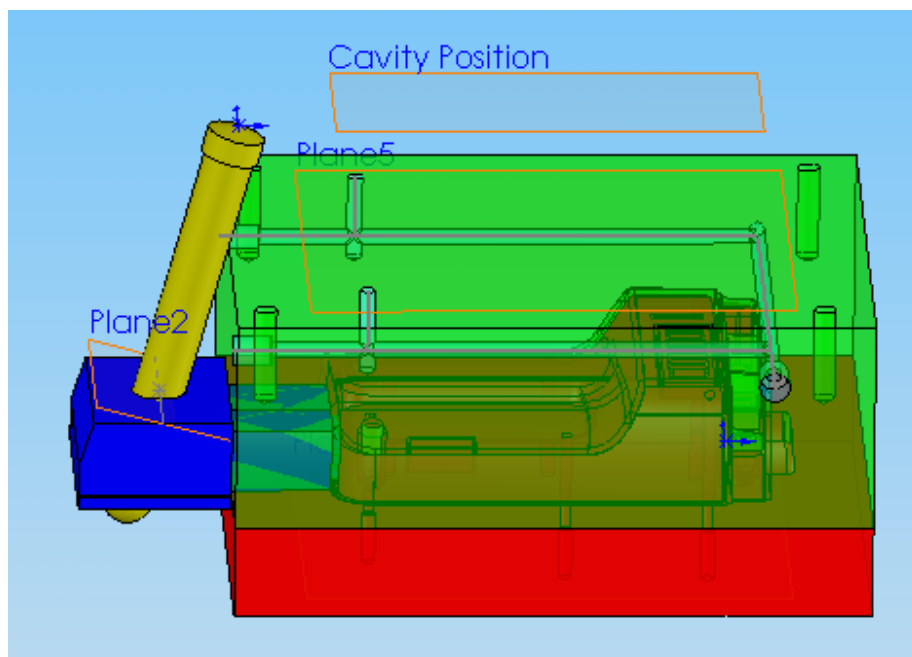
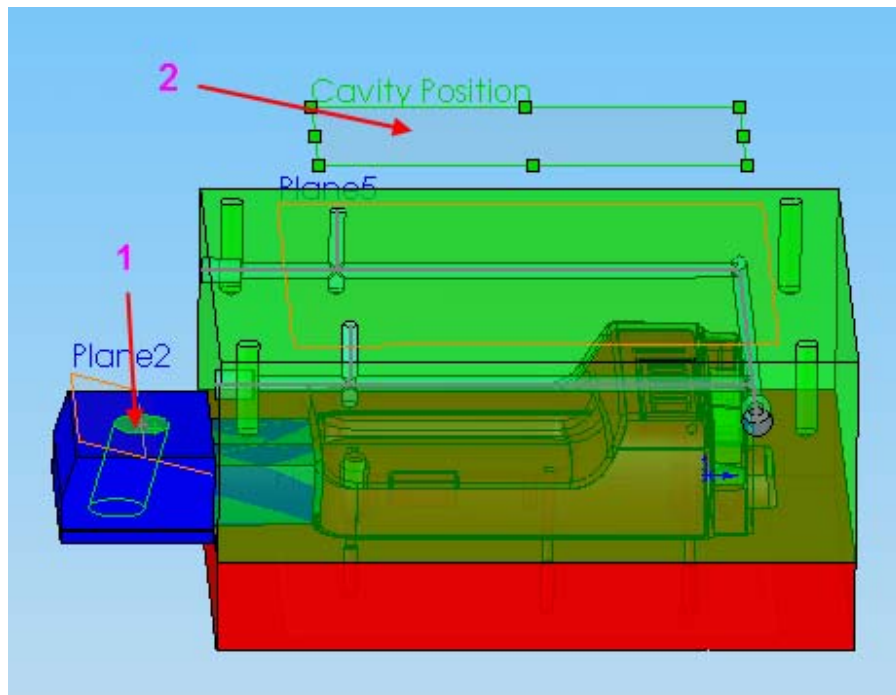


The following icons are used on assembly file.



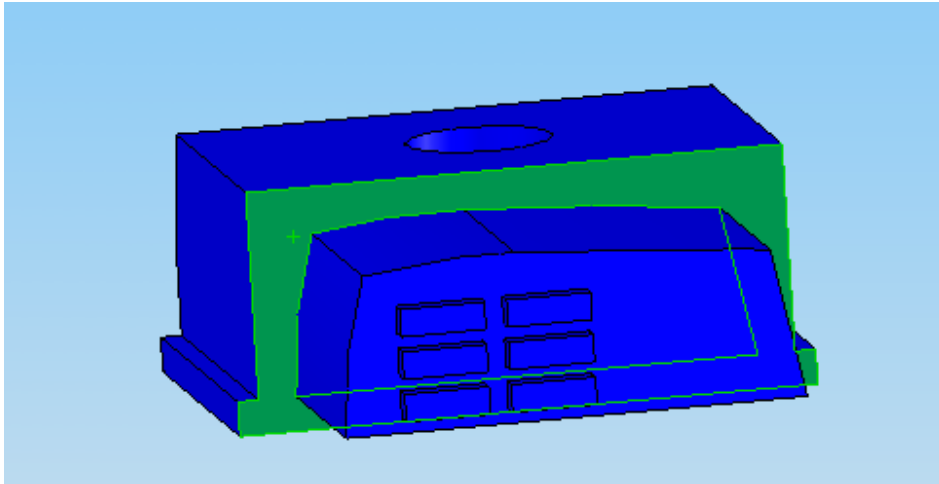
: Add angular pin for slider

Method: select the cylindrical face and a reference plane in sequence as follows, click the icon.

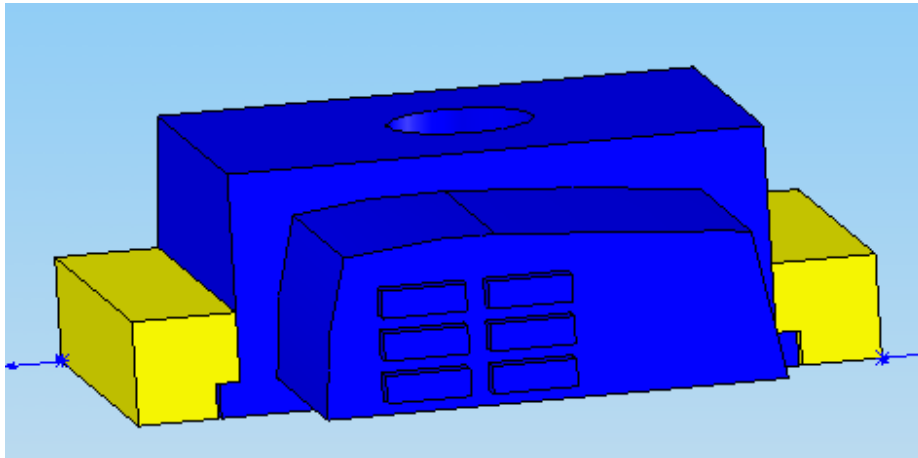


: Add guide rail type 1

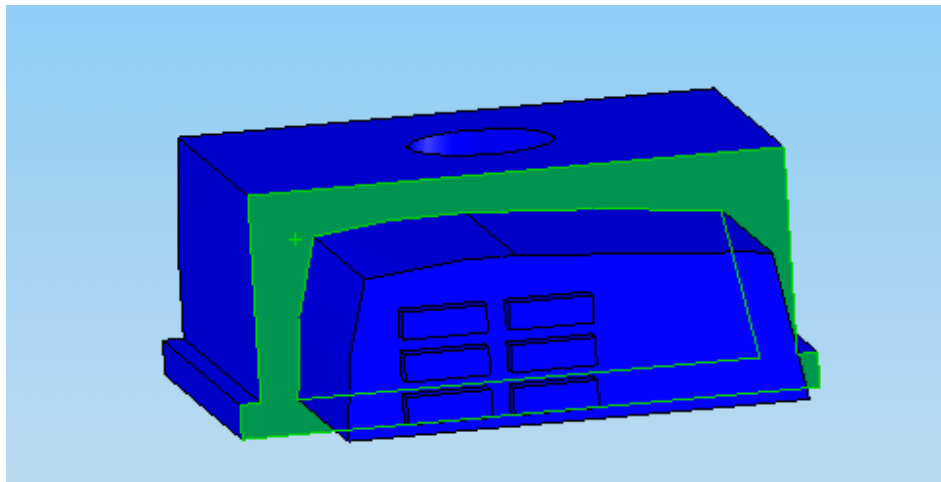
Method: select the face as shown below, click the icon.

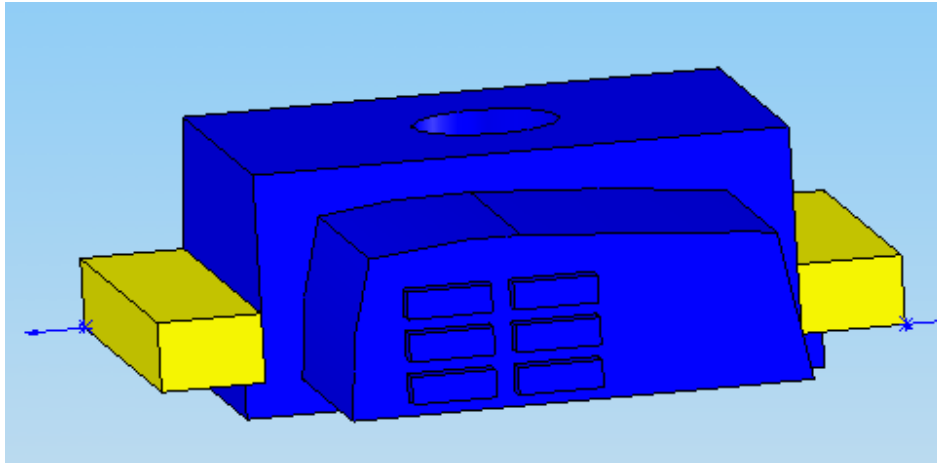


result



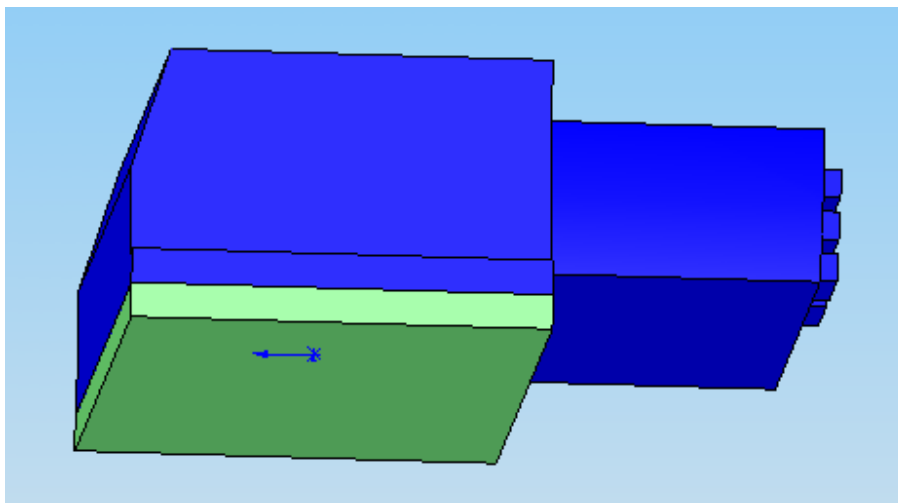
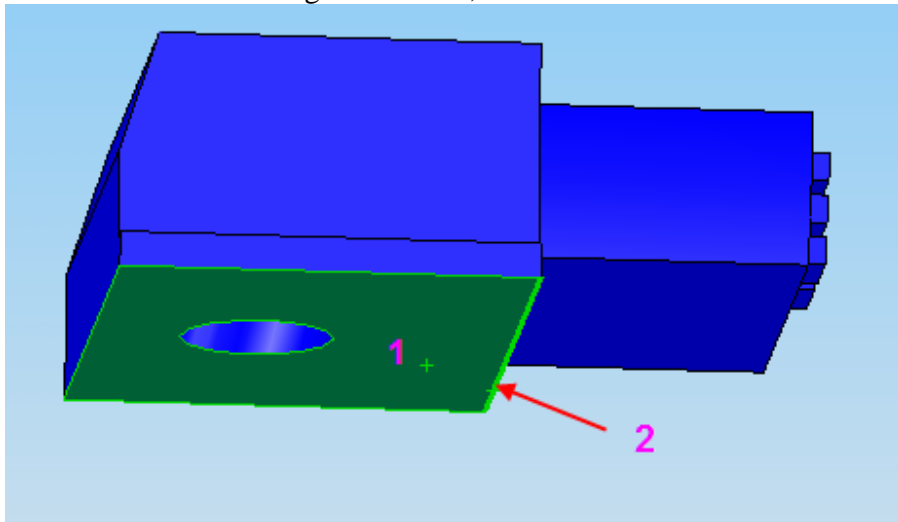
: Add guide rail type 2
Method: Same as the above





: Add wear plate

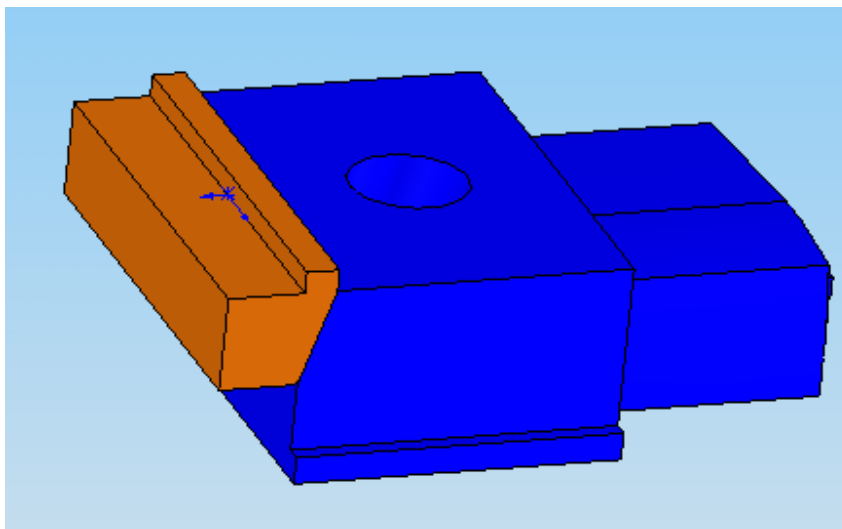
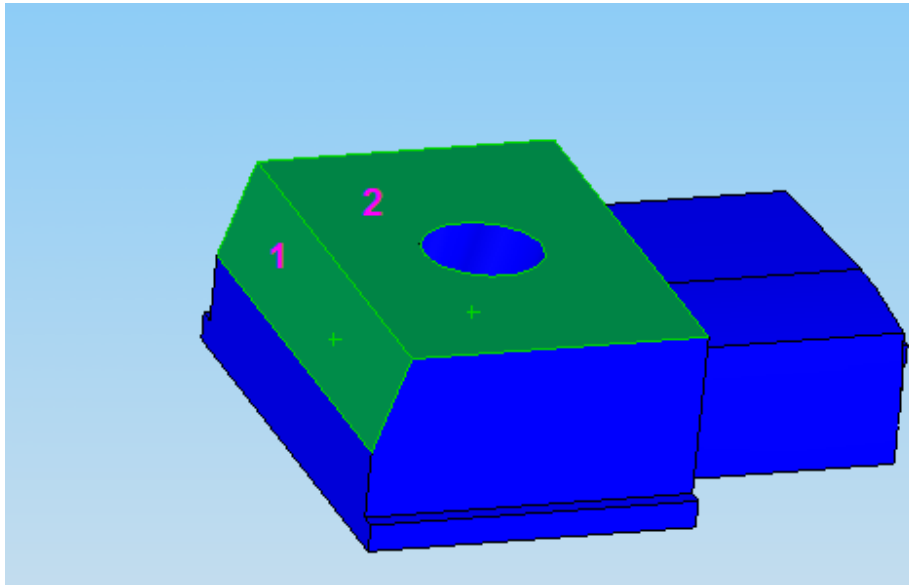
Method: select a face and an edge as follows, click the icon.





: Add locking block type 1

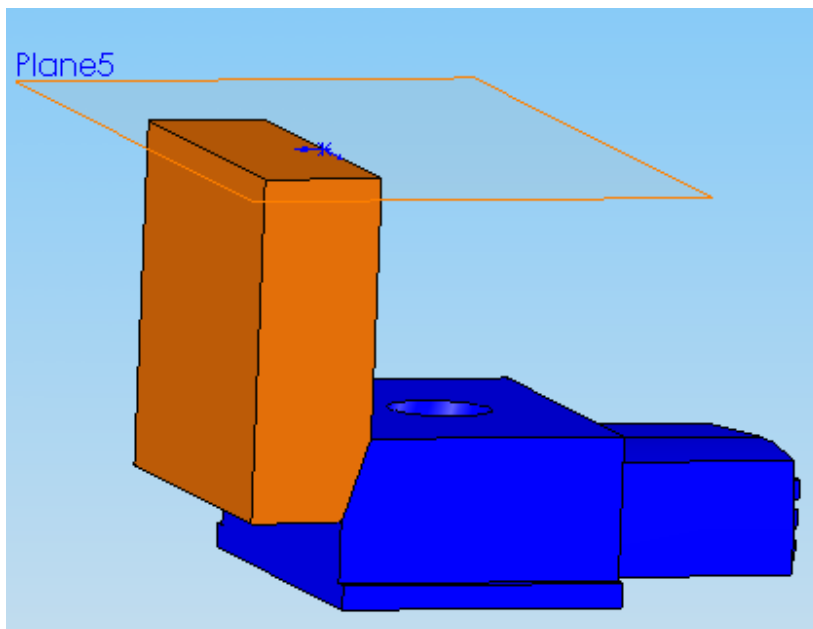
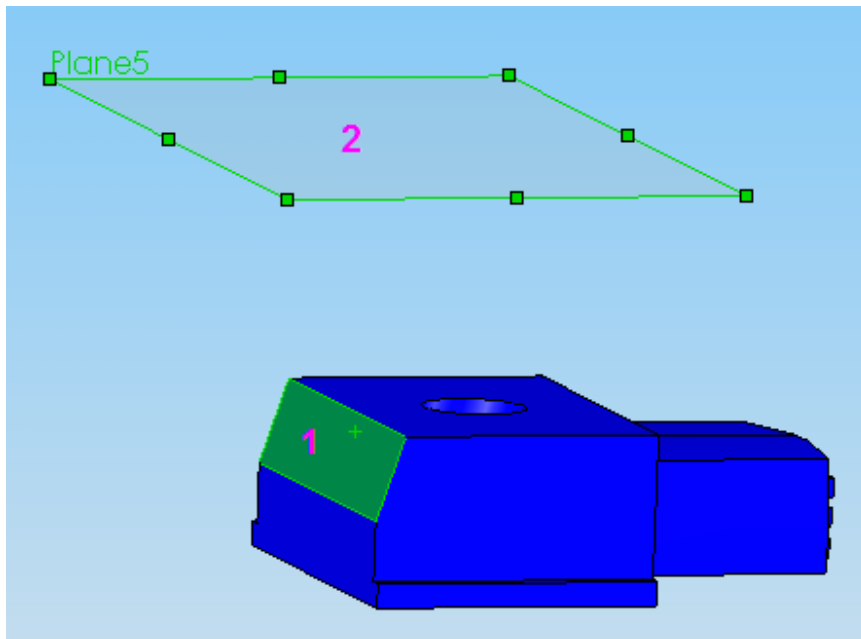
Method: Select 2 faces in sequence as follows.





: Add locking block type 2

Method: Select a face and a reference plane in sequence as follows.



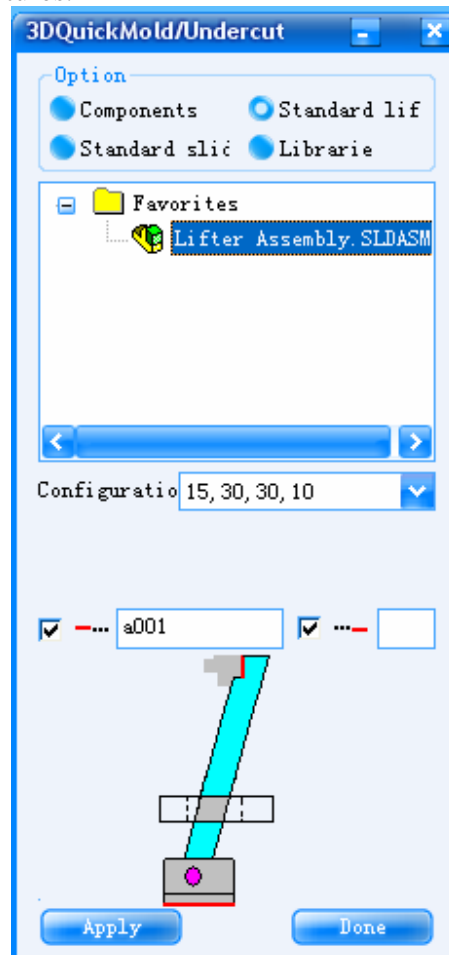
Not available at moment



Not available at moment

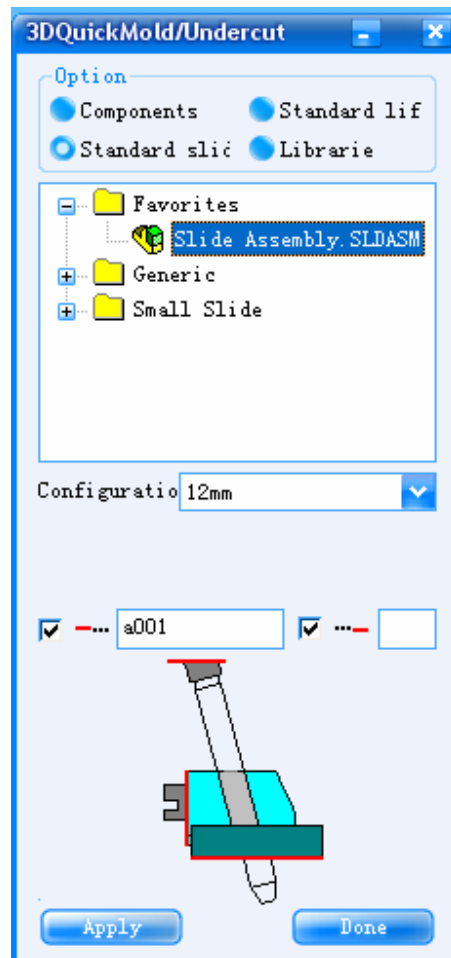
2. Standard lifter

Some standard lifter structures.



3. Standard slide

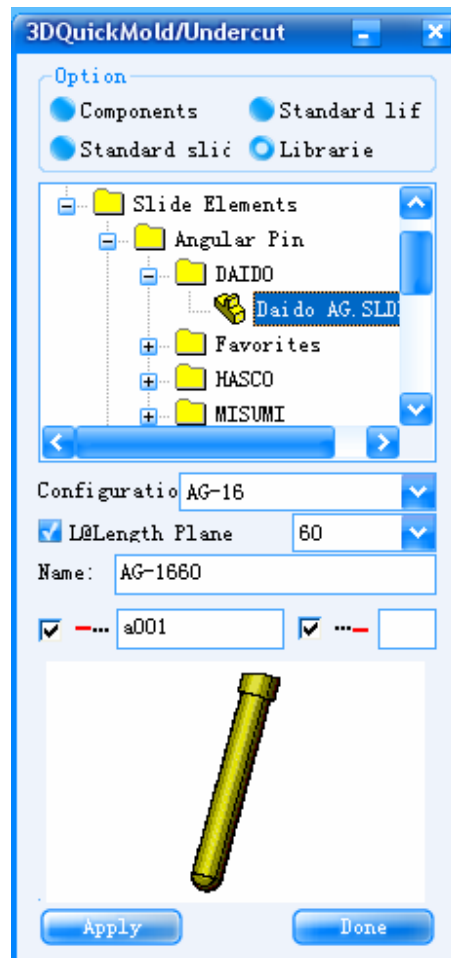
Some standard slide structures.



4.Libraries

Some standard components for slide and lifter.

Method: import the part, place it to a suitable position.




11.SubInsert Manager

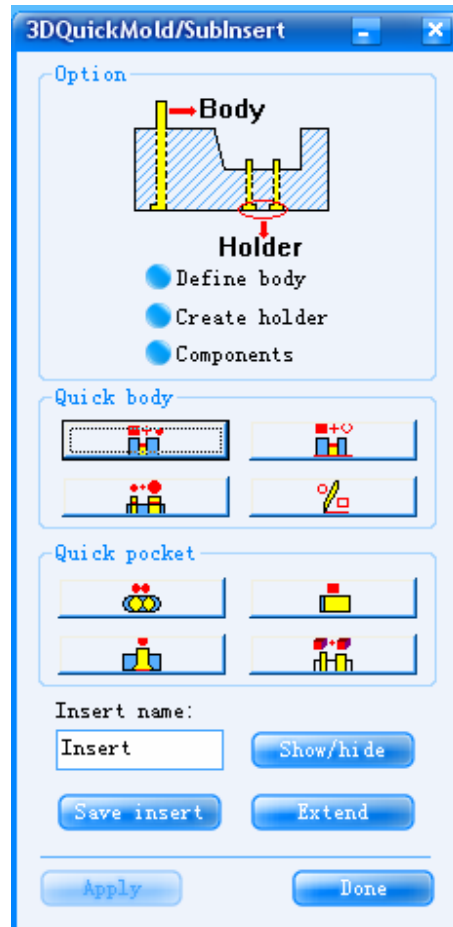
Usually, there are two methods for sub-insert design

Method 1. Design sub-insert on product, this step is done before mold splitting

Method 2. Design sub-insert on core or cavity, it is done after mold splitting

Sub-insert Manager is mainly used for the above method 2.

Click icon , the following pop outs



Typical design procedures as follows:

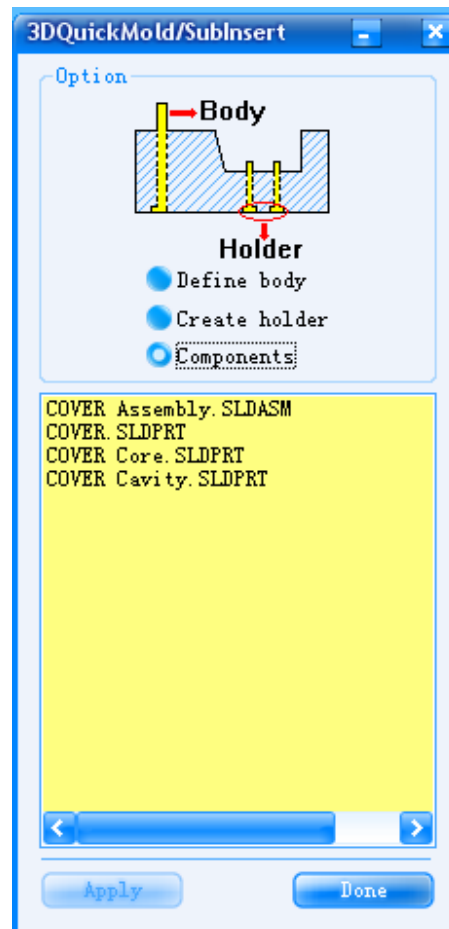
Define sub-insert body → Create holder → Save sub-insert

For Define body and Create holder, select the option, click Apply to enter the the property manager page. Click Done to quit dialog.

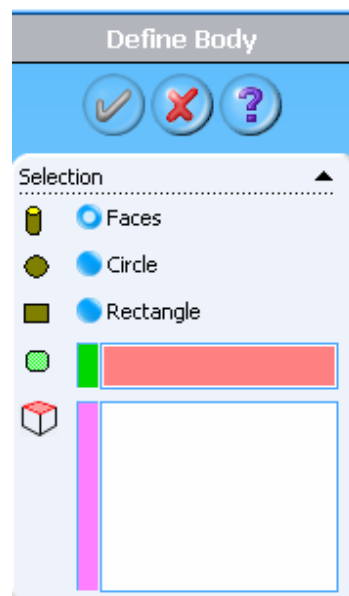
The icons on Quick body are used to quickly generate the Sub-inserts body. Before using this icons, some pre-selections are needed.


1. Components


In component list, all sub-insert design related files are listed, double click on the parts or assemblies to activate the working document.

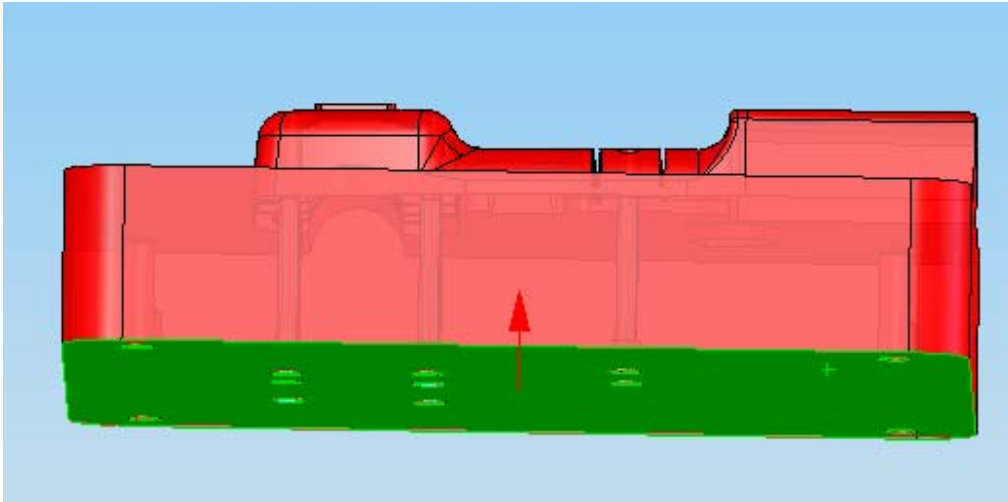



2. Define body

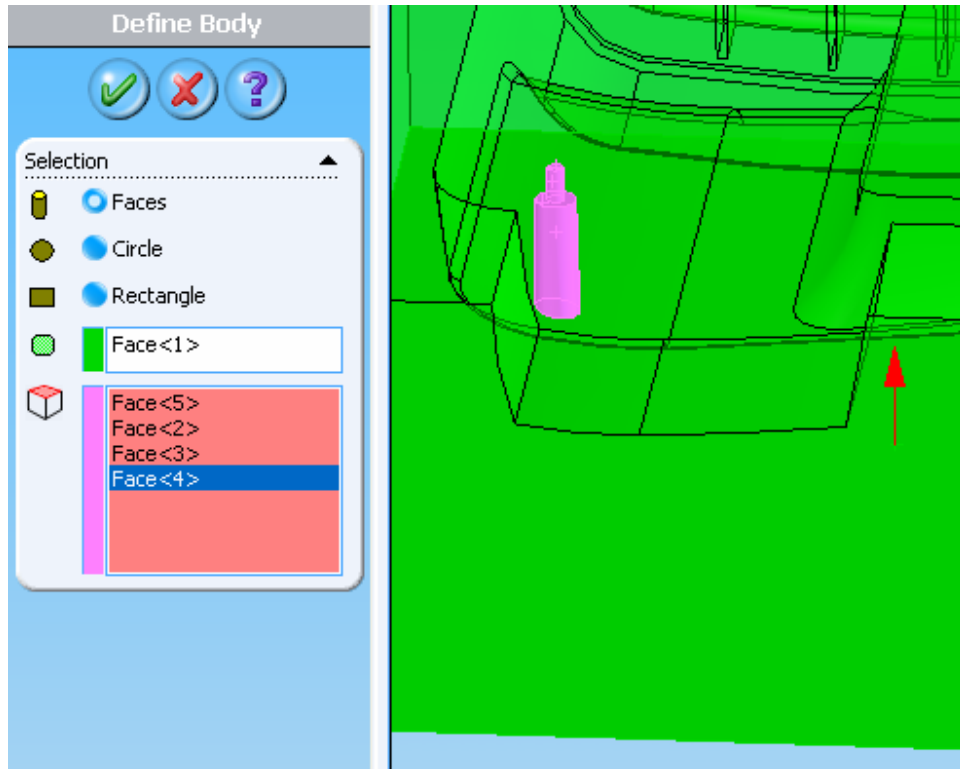


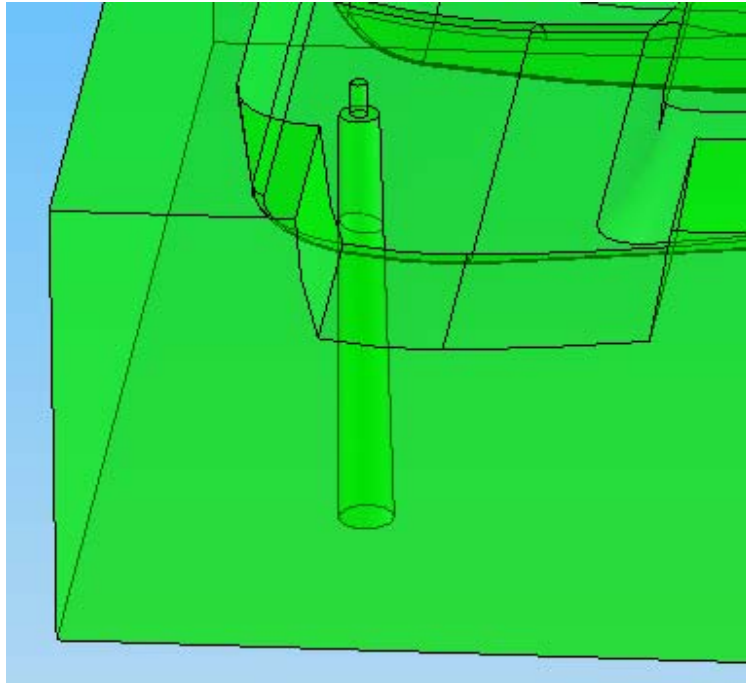
 Faces: Define the sub-insert body using selected faces

: select a plane to be the reference plane for the sketch. 3DQuickMold will automatically select the bottom face as the reference plane.

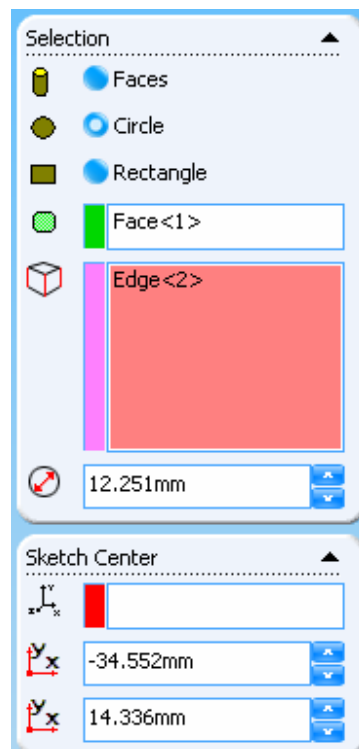


: Select the face to build the sub-Insert.






- Circle: create a circular sub-insert





: select edges and vertex as the reference to define the circular sub-insert body

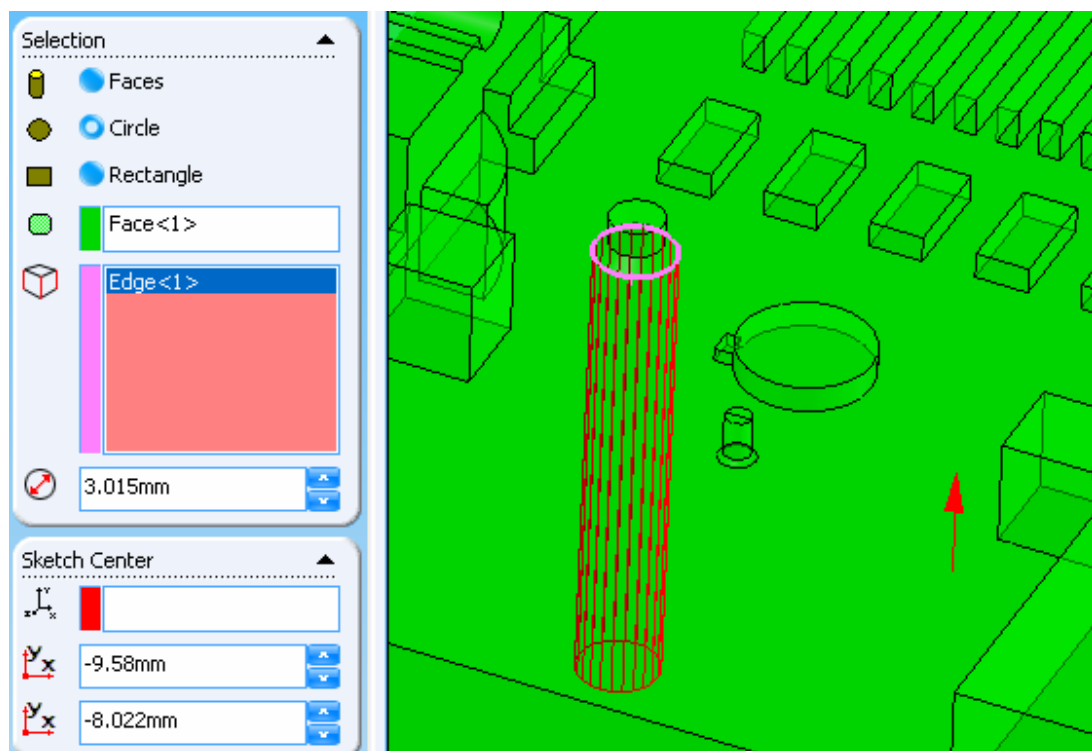
The selected entities are used to define the minimum circular profile


: the diameter for the circular sub-insert

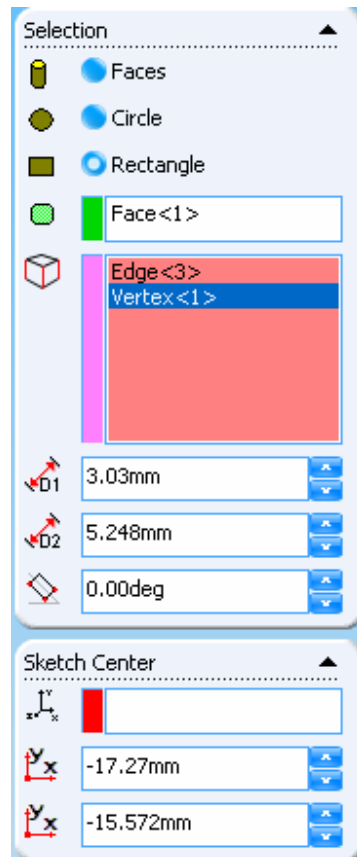
Sketch Center


: Select coordinate

: Display the position of the center of the Subinsert relative to the coordinate system. If the coordinate system is not selected, the part coordinate system is selected. The center of the sub-insert can be changed by changing the setting here.



 Rectangle: create a rectangular sub-insert




: select edges and vertice as the reference to define the rectangular sub-insert body


The selected entities are used to define the minimum rectangular profile


D1: define the length of one side

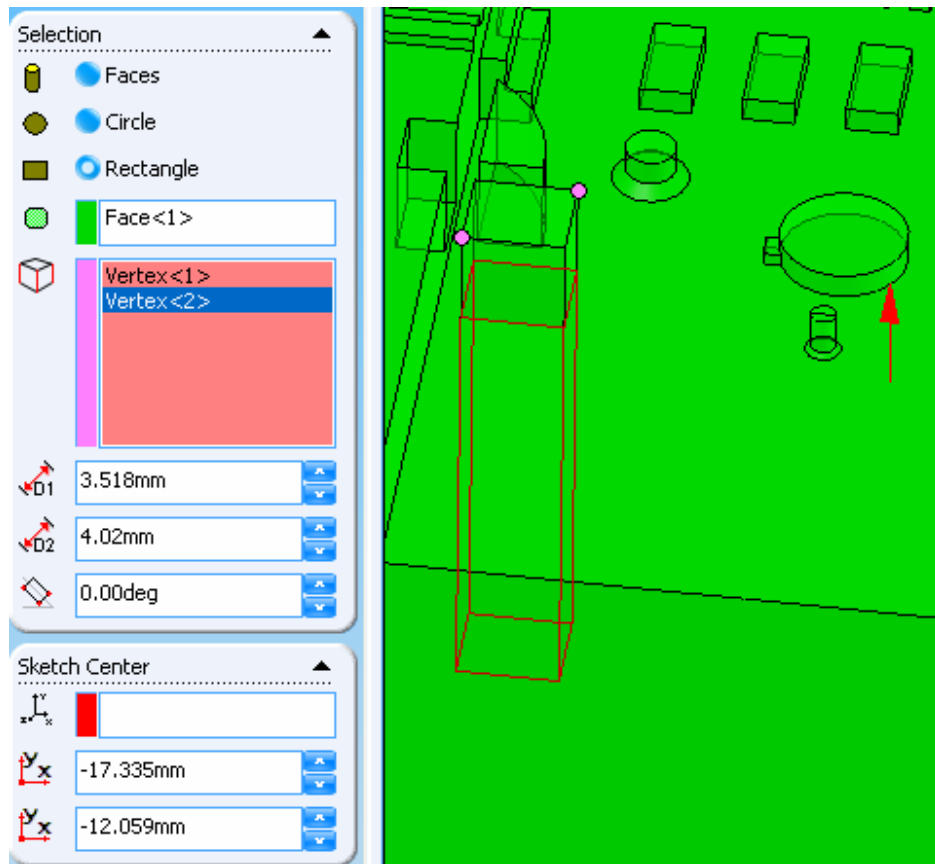
D2: define the length of anther side

: define the angle of rotation of the sketch

Sketch Center

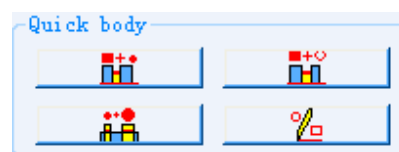
: Select the coordinate system

 Display the position of the center of the sub-insert relative to the coordinate system. If the coordinate system is not selected, the original part coordiate system is selected. The center of the sub-insert can be changed by changing the setting here.



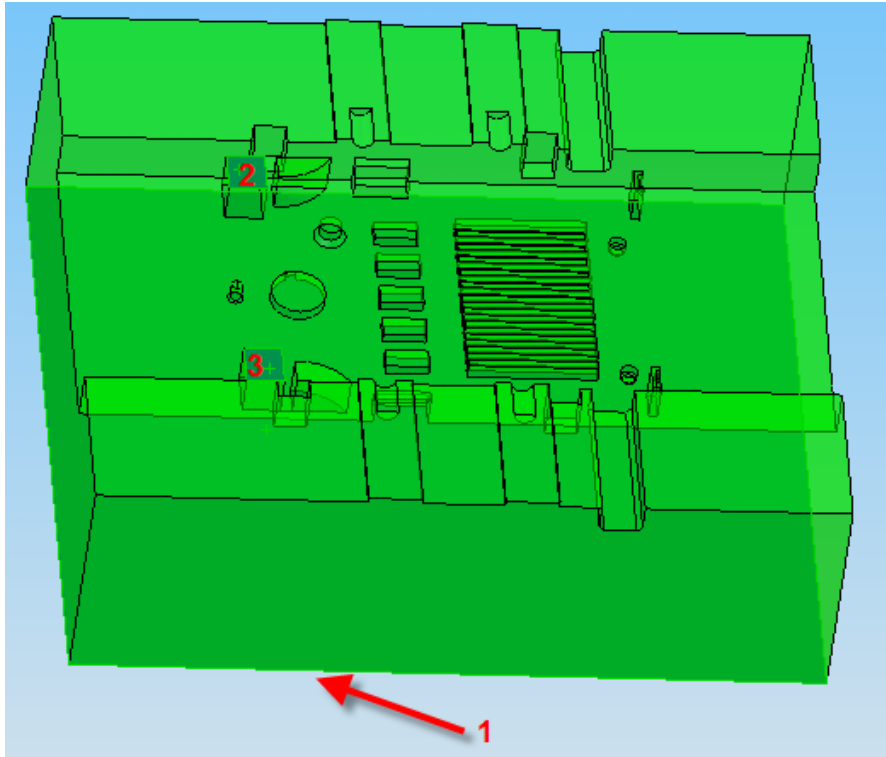
3. Quick body

The functions here require pre-selection. They are used to create sub-insert body effectively, using those, no PMP style interface will pop up.

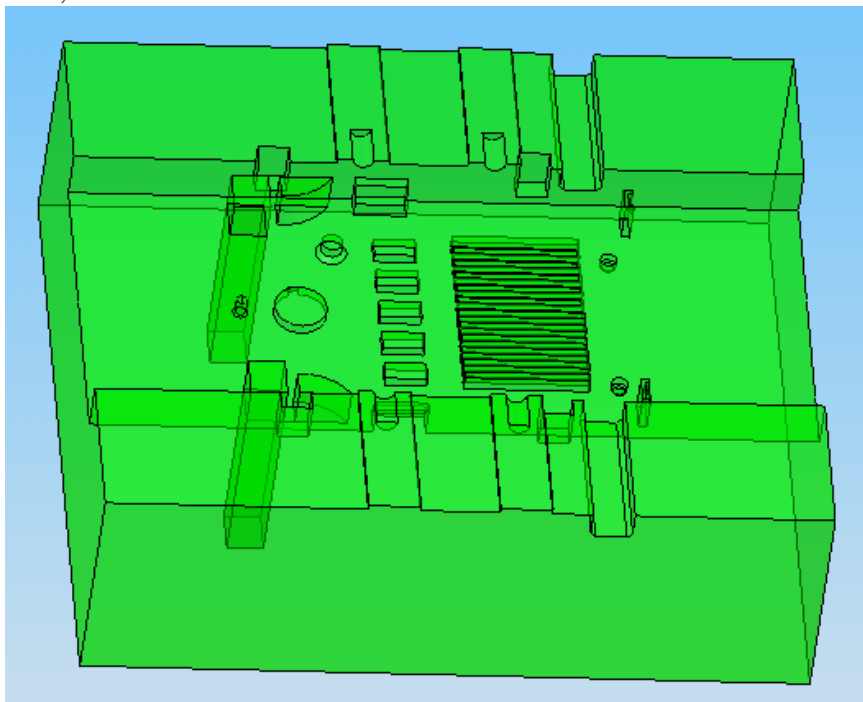


: Select in sequence the bottom face of the mold core or cavity and some faces for building the subinsert.

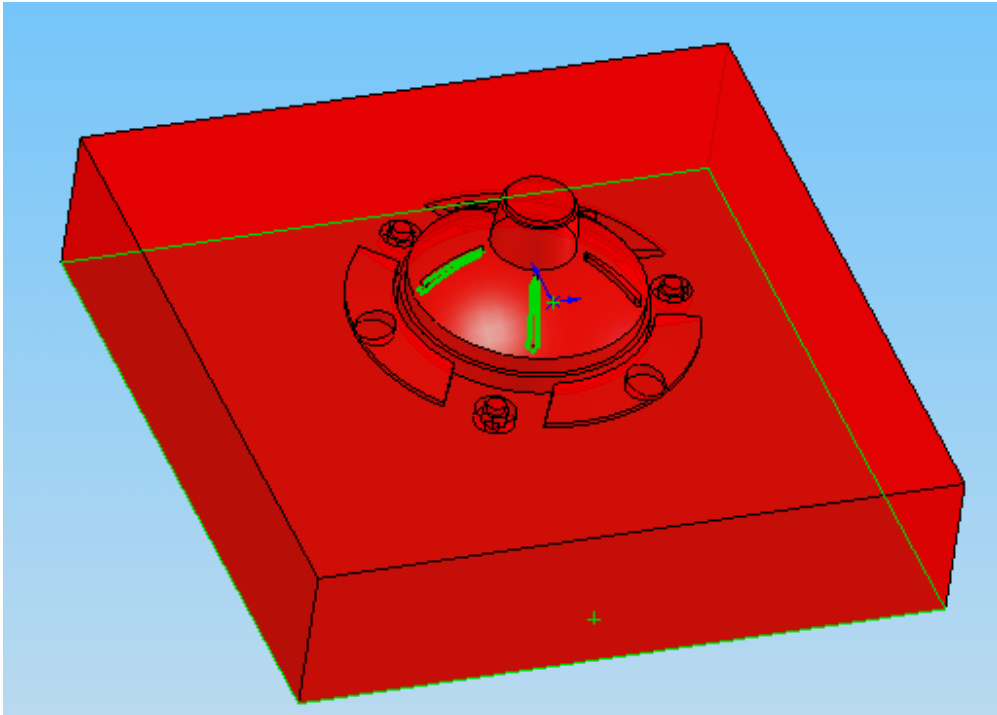
Follow the order to select the bottom surface of the mold core cavity and the required (one or more) faces to define the sub-insert



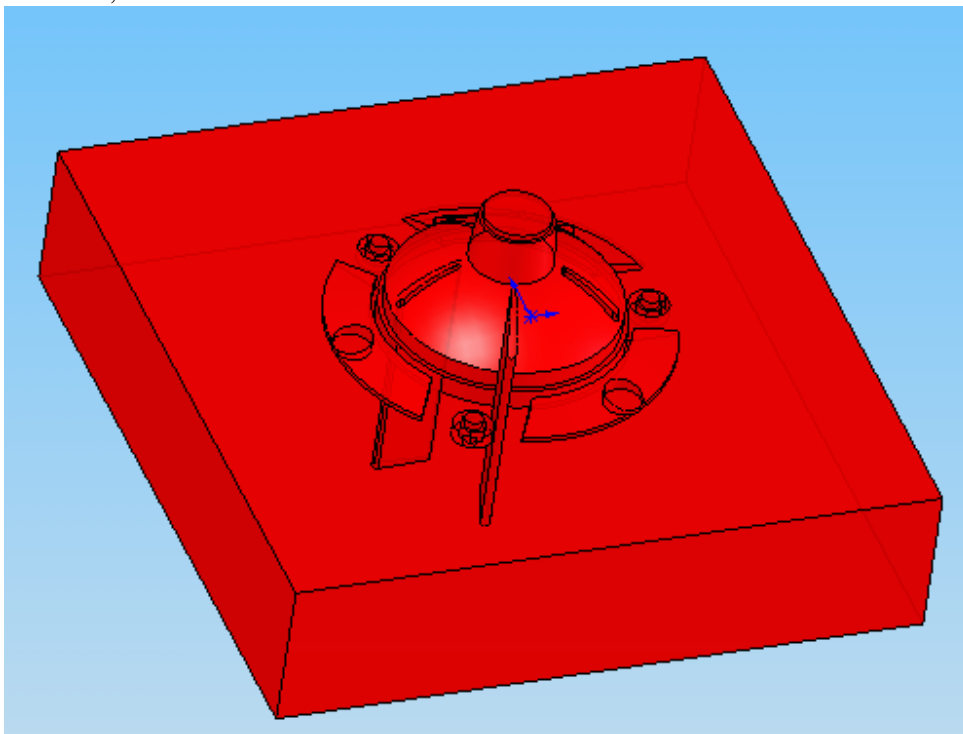
Click this icon, and the result is shown



: Select in sequence the bottom face of the mold core and cavity and a closed chain edges to define the sub-insert body.

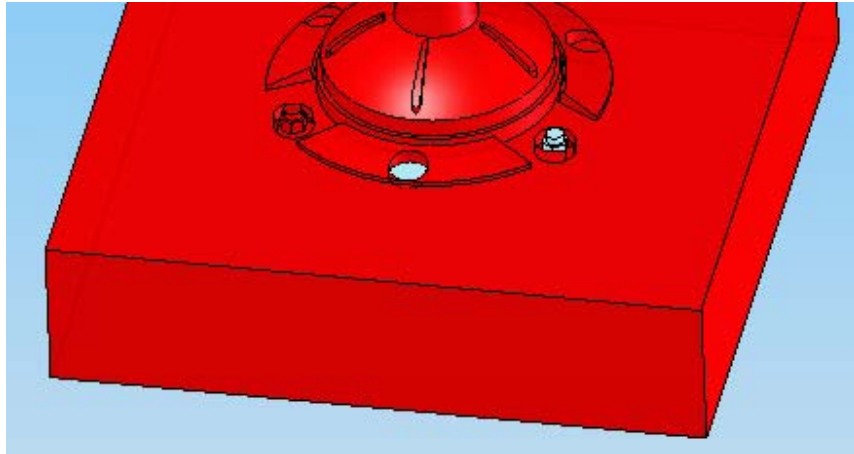


Click this icon, the result is shown



: This function is particularly used to combine with the “Paste Body” operation.

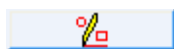
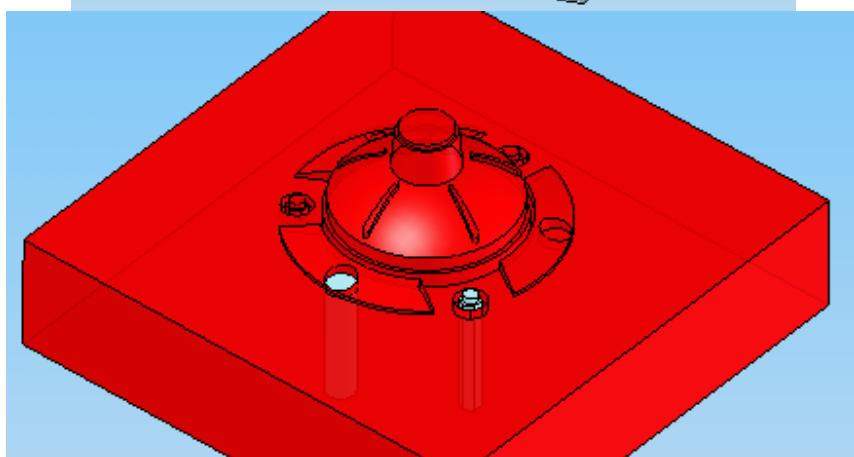
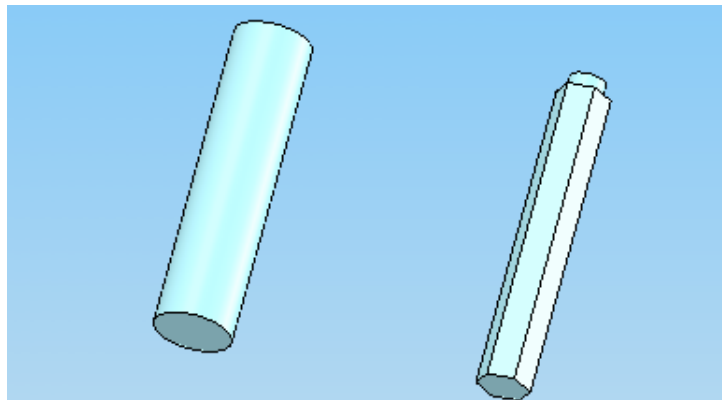
For example, the blue body shown below is created by the Past Body



Select the face of Body facing the bottom face of the mold core

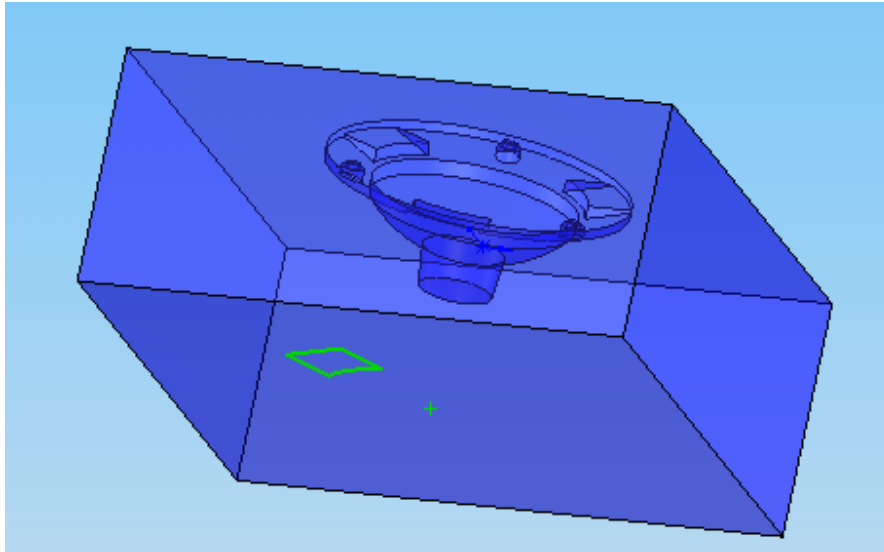


Click the icon

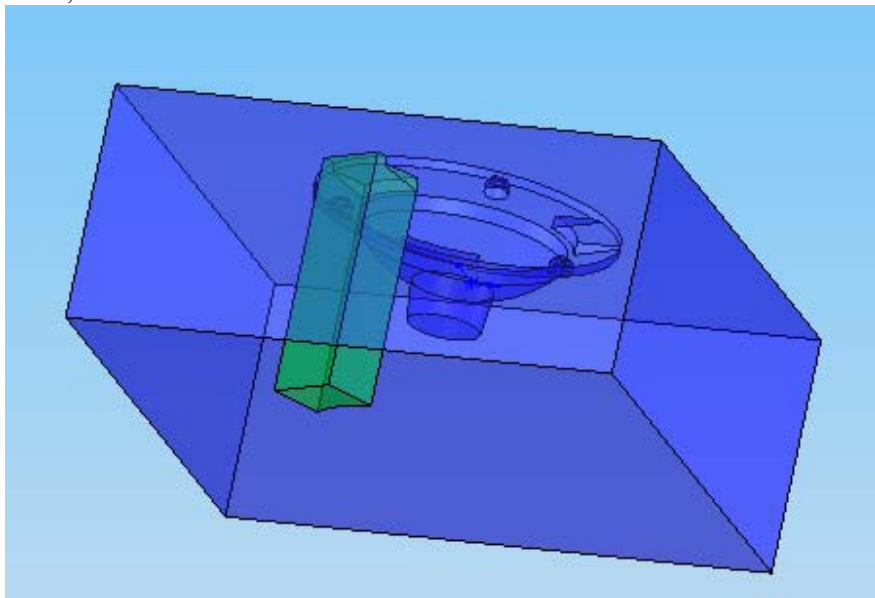


: The most flexible method to create sub-insert is to use pre-defined sketch, this sketch should be created on the bottom face of core or cavity.

select the sketch on the bottom as shown



Click this icon, the result is shown

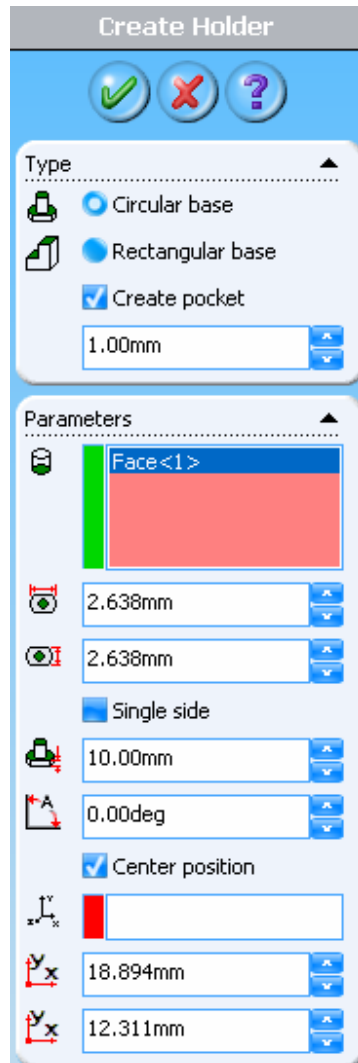



4. Create holder


Select “Create Holder” then “Apply” to build Subinsert holder.

Under type, circular and rectangular type are available.


Create Pocket : Make this option checked, a pocket will be created as well to fit the created subinsert. The number below affects the value of the gap between the sub-insert and the pocket.



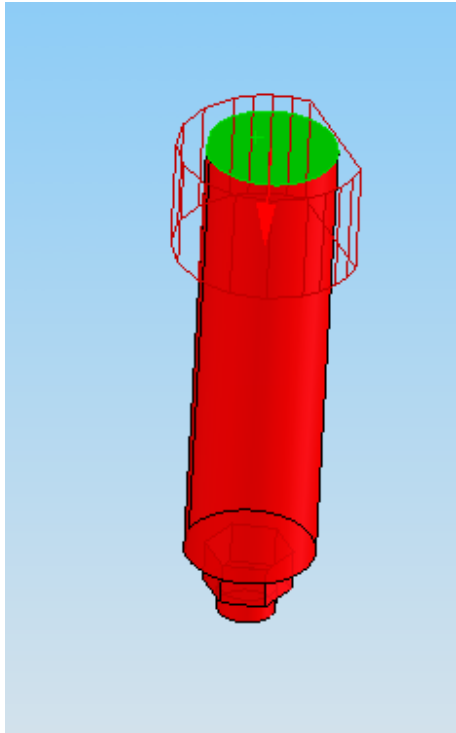
 **Circular base:** Used to add a circular holder. For circular holder, multiple bottom faces can be selected at a time. The default dimension is the smallest dimension.


 : Select a bottom planar face on the sub-insert to define the holder sketch.


 : Define the Diameter of the holder.

 : Define the distance if the desired bottom shape is required, for round one, set this value equal to the diameter.


Single side: The above cutting is on one side if this option is checked. The preview as following picture shown.





: Define the height of the holder


: Define the angle of rotation

Center Position: By default, the holder sketch is located at the center of the selected bottom face, when this option is selected, its position could be specified.

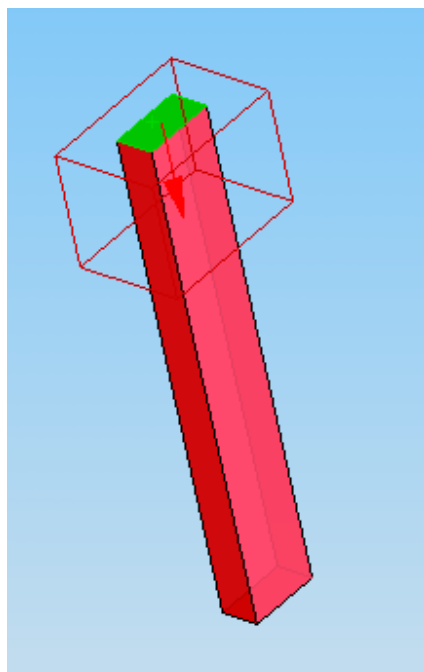
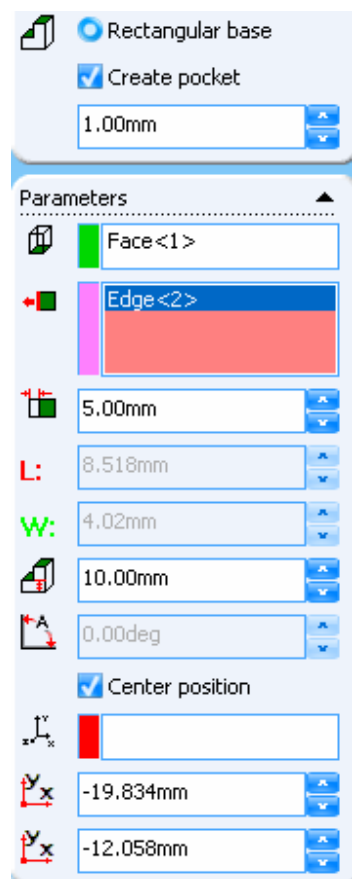
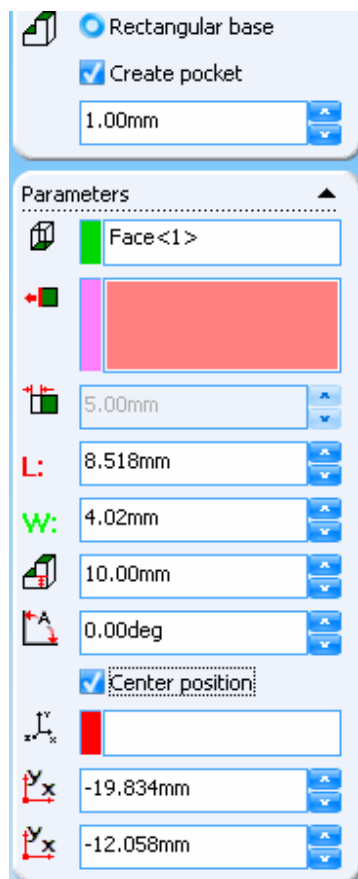
: It is for selecting the coordinate system

: Display the position of the center of the Subinsert relative to the above selected coordinate system. If the coordinate system is not selected, the coordinate system is the part's one. The center of the Subinsert can be changed by changing value.

 **Rectangular base:** Used to add a rectangular holder.


: Select a bottom face on the sub-insert to define the holder.


: Select one or two edge on the bottom face as the references to define the offset on the holder. If no edge is selected, 3DQuickMold will define the holder by dimensions.





No edge is not seleted

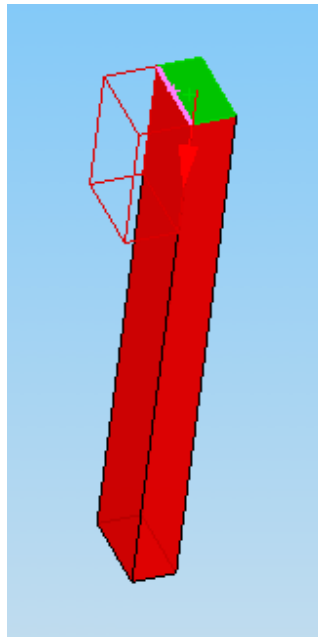
Set the following dimension to define the holder size.

 : Define the length of the holder

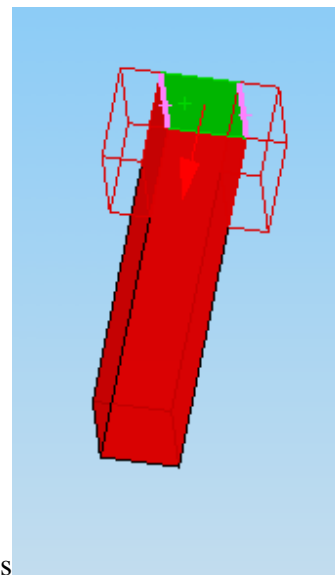
 : Define the width of the holder

 : Define the height of the holder

 : Define the angle of rotation




Select one edge




Select two edges


 : If one or more edge is selected, set the following dimensions

 : Offset value

 : Define the height of the holder

Center Position: Used to define the center position

 : It is for selecting coordinate system

 : Display the position of the center of the Subinsert relative to the above selected coordinate system. If the coordinate system is not selected, the coordinate system is the part's one. The center of the Subinsert can be changed.

5. Quick Pocket



6. 其他 other

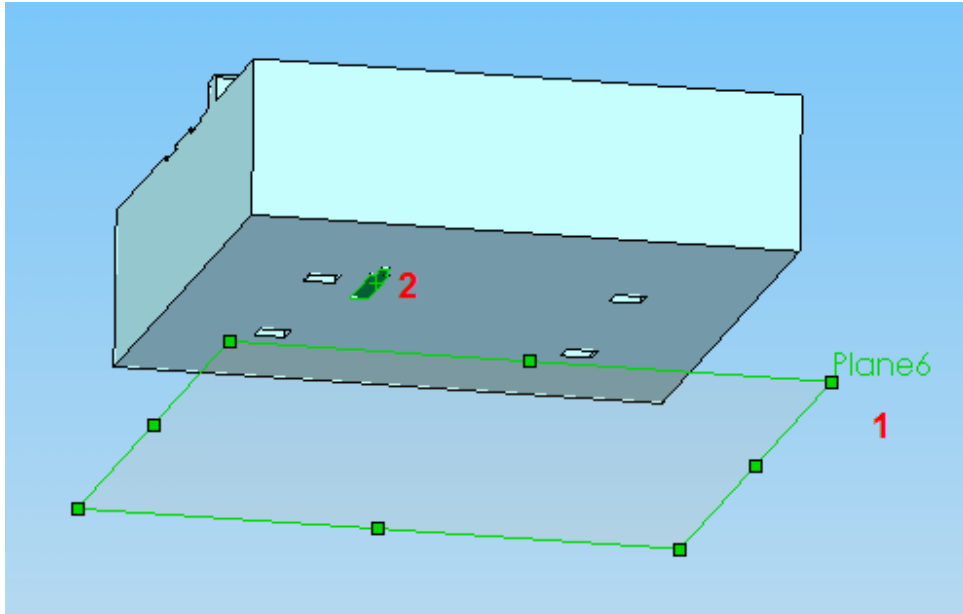
Insert name: Prefix for the sub-insert to be saved.

Show/hide: Show/hide the sub-insert's bodies. For example, on mold cavity, click this icon to hide all bodies but the main body, click this icon again, all hidden bodies will be shown again.

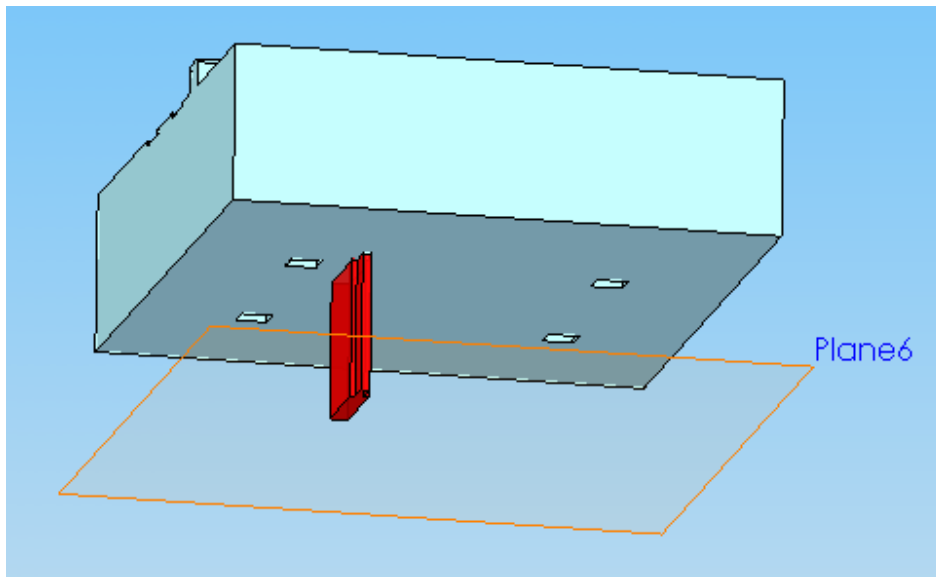
Save insert: Save the sub-insert to a separated component and insert it the product assembly. This function can be used on both part file and assembly file. To run this function, you need to select a face on the sub-insert.

Extend: Extend the bottom face on the sub-insert to a particular plane, user needs to select one reference plane and one bottom face of the sub-insert in sequence, and click the icon. It is normally used to extend the sub-insert to the bottom plane of cavity plate or core plate.

As shown belows, select a plane and a bottom face of the sub-insert



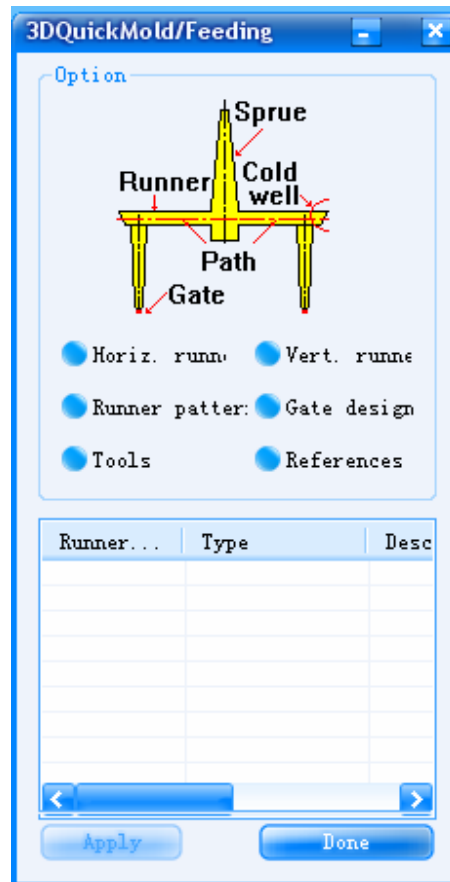
Click Extend



12.Feed Manager

Feed Manager is for runner, gate design, usually it is used after Layout. In the option of Layout, creation of a * Runner.sldprt file in * Project.sldasm is set as default, and this file is used for runner design

Click  on the main toolbar to start feed manager.

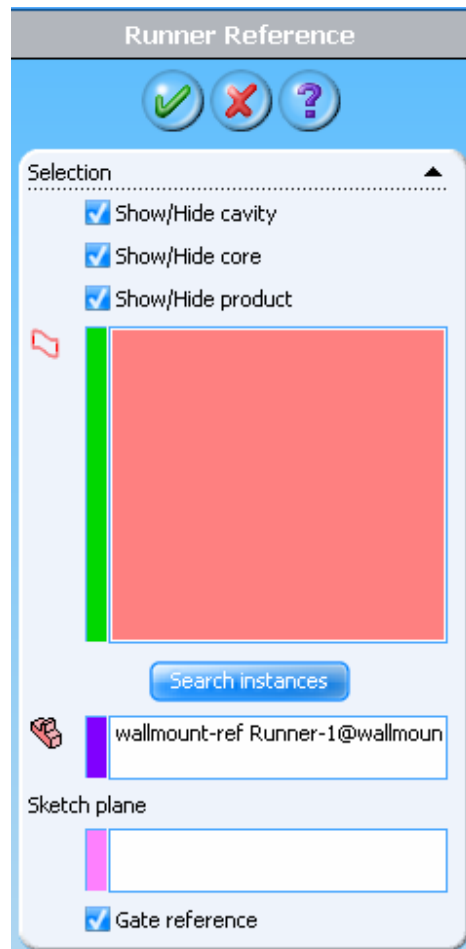


Note: there are 6 options in Option group, References can only be used on assembly file, and the other 5 options can only be used on part file.

1.References

3DQuickMold doesn't create feed system at the assembly environment directly, instead, it designs the runner and gate on part file which is prepared when the layout was created.

Reference helps to add the faces from the assembly file that is related to define the position of the runner to the runner file. The runner is designed then. This method helps to convert the work done from assembly to part; it can improve the design performance greatly.



Show/hide cavity: Show or hide cavity in the current assembly. This option will be disabled when **Reference faces and each instances** are selected.

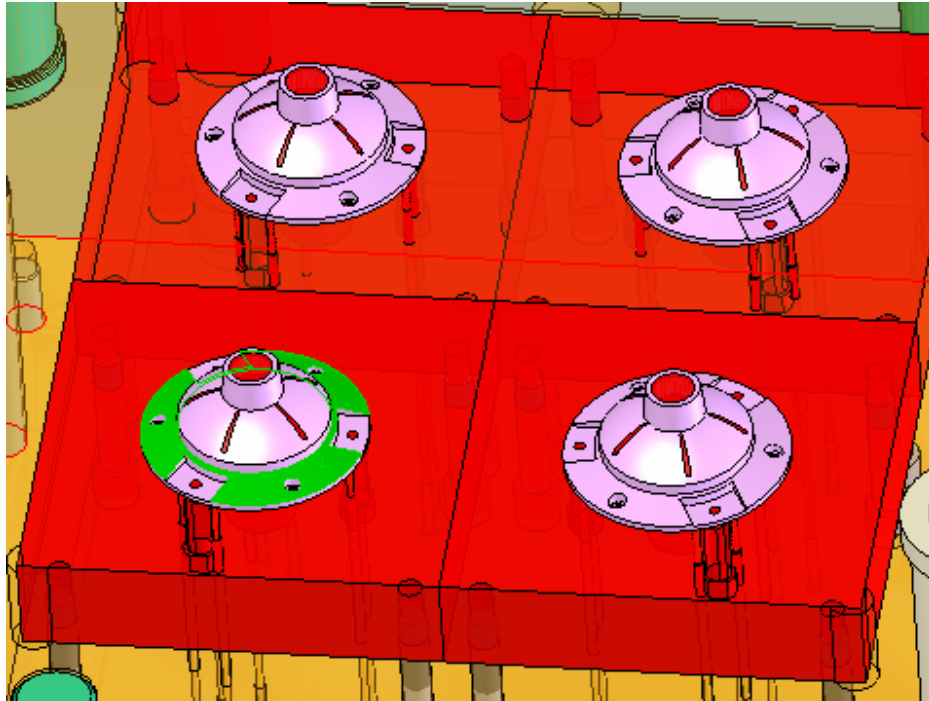
Show/hide core: Show or hide core in the current assembly.

Show/hide product: Show or hide product in the current assembly.

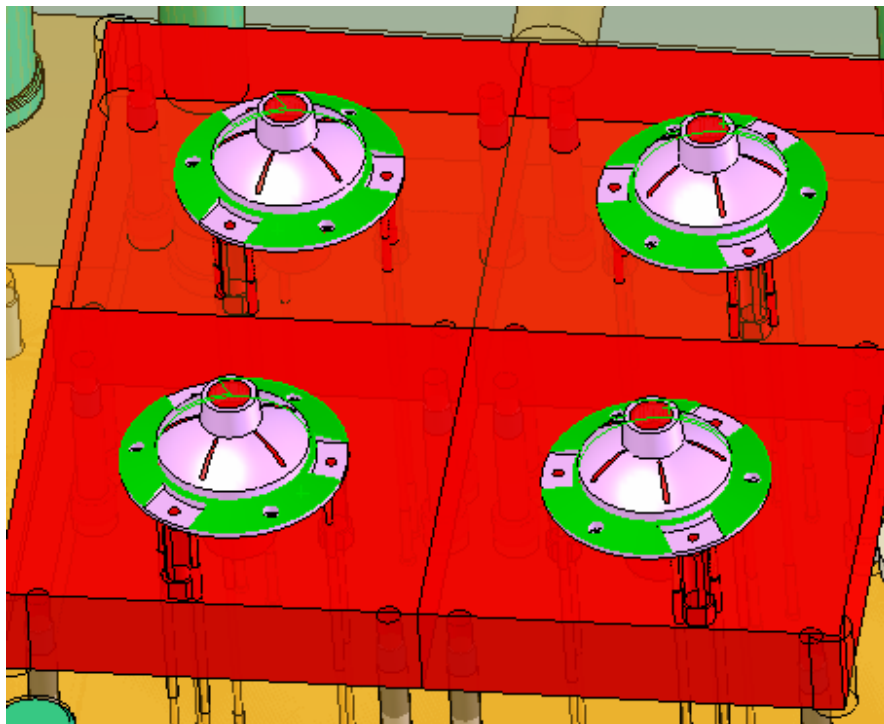
Reference faces: select the face to be the reference face for the runner design

Search instances: for multi-cavity layout, select a face on product/core/cavity, click this icon, 3DQuickMold will automatically add the same face from the other cavities to Reference faces

Select a face on one product file as the following pictures shows

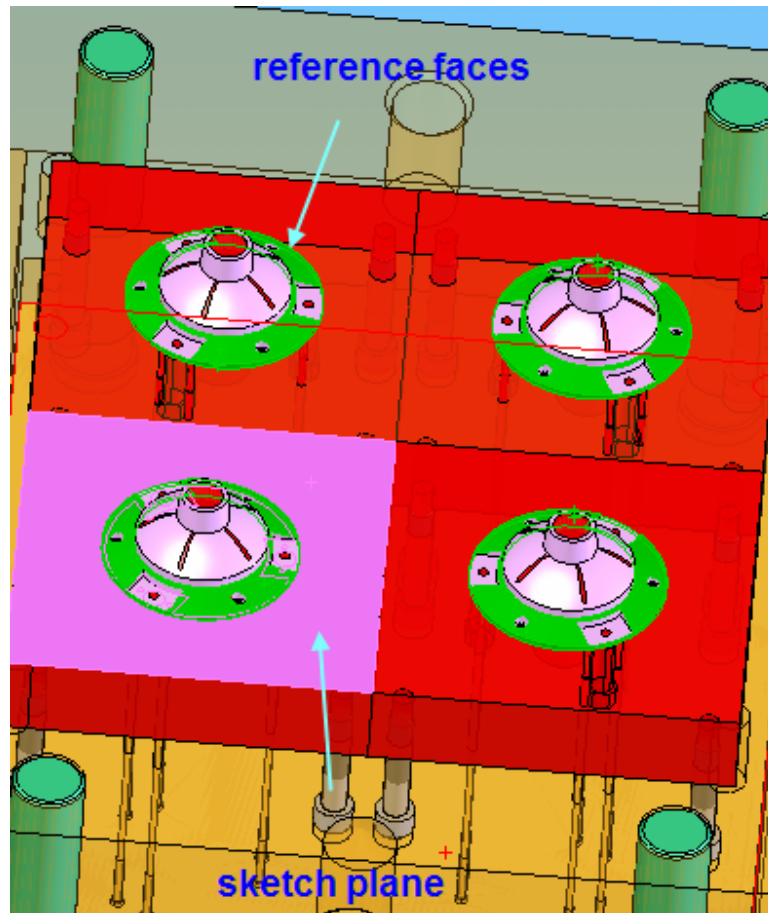


Click Search instances

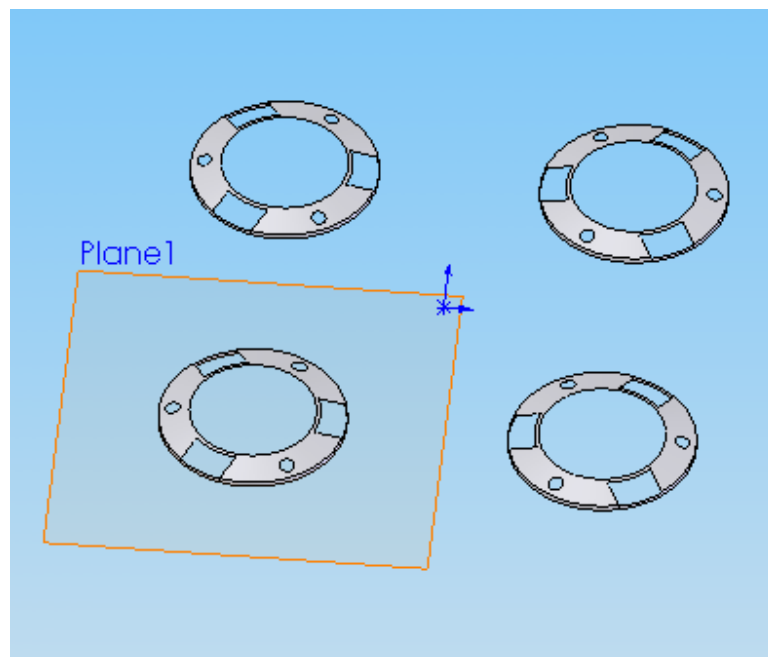


Runner part: runner will be saved as a single part, input the name of the runner part here

Sketch plane: Select a plane as reference for the main runner creation.

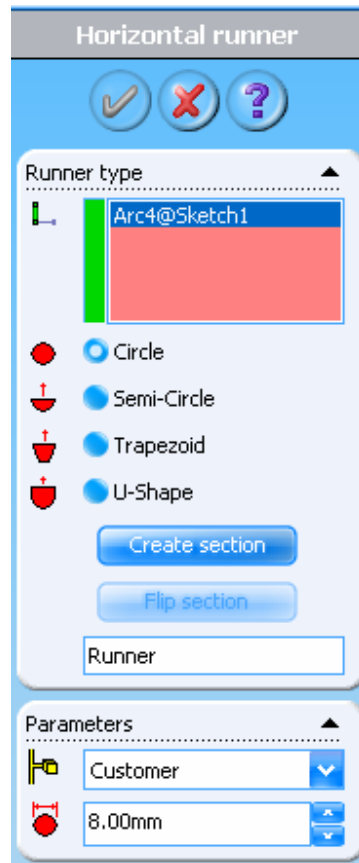



Click OK, then open * runner.sldprt, the result as follows








2.Horizontal runner





Design runner that is parallel to the PL(parting line) face.




 : Select sketch segments to define the runner path, segments selected should not be crossed and must be continuous. If many runners have to be created, create the runner one by one;



The shapes of runner are categorized as follows:


1.  : select circle as cross-section of the runner;
In Parameters, , define the diameter for the runner
2.  : select semi-circle as the cross-section of the runner ;
In the Parameters, , define the radius for the runner
3.  : select trapezium as the cross-section of the runner;
In Parameters,

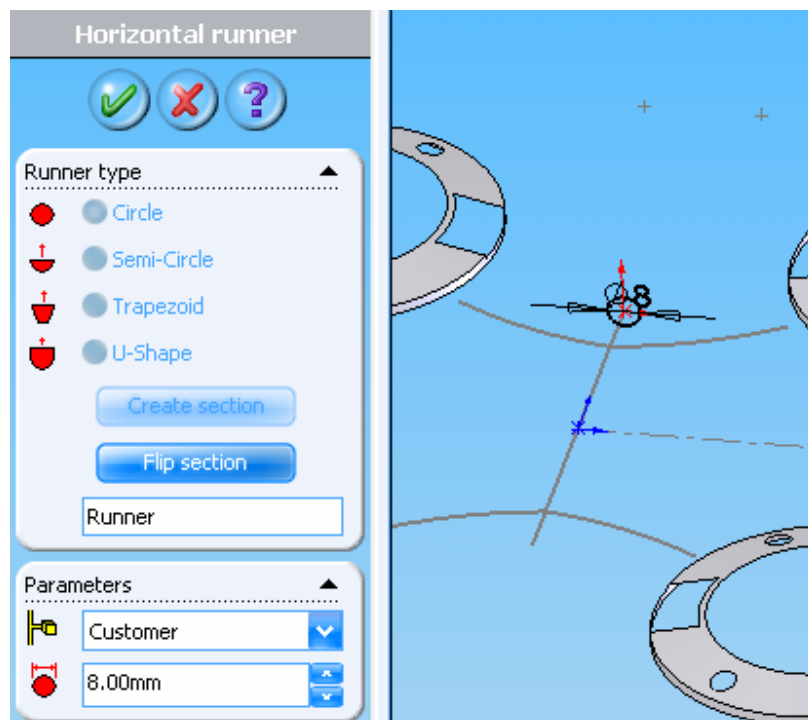
- , define the width of the top face of the trapezium;
- , define the depth of the trapezium;
- , define the taper angle of the trapezoid;
- , define the corner radius for the bottom of the trapezium;

4.  : select U-shaped as the cross-section of the runner;

In Parameters:

- , define the radius for the U-shape runner;
- , define the taper angle of the U-shape ;

Create section: After selecting the runner path and the cross-section of the horizontal runner, click this button, a sketch will be created. The  selection will be hidden, "Flip section" button will be activated;

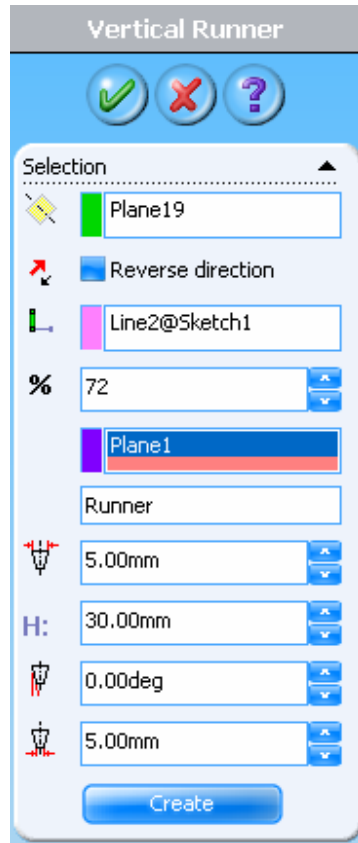


Flip section: Adjust the direction of the cross-section of runner. Click the function and the sketch will rotate by 180°


Runner naming prefix: Name the runner.


3. Vertical runner

Used to design the runner that is perpendicular to the PL surface, for example main runner.



: select a reference plane to be the sketch plane of the vertical runner, an arrow will appear and indicate the direction for the vertical runner


: If the direction is not correct, click this function to flip the direction;

: Pick up a point to be the centre of the vertical runner sketch; sketch point or sketch segment can be selected, if an segment is selected, % will be highlighted


%: Adjust the the position ratio of a point on the selected segment.


End Reference: The ending surface of the runner

Runner naming prefix: Name the runner

: define the diameter of the vertical runner;

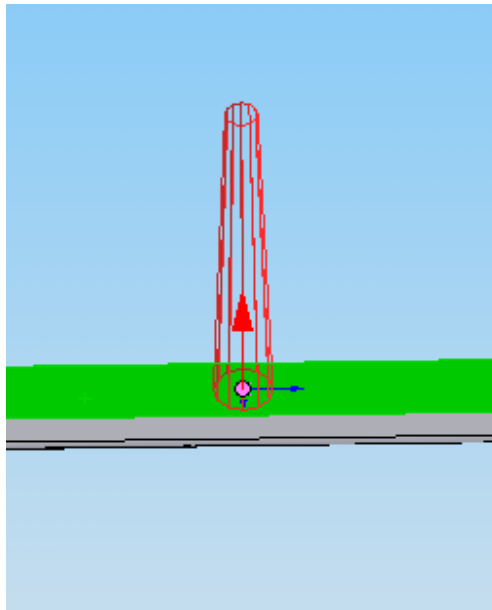
H: define the height of the vertical runner;

 : define the taper angle of the vertical runner

 : define the diameter of the other side of the runner;

For the latter four setting, complete any 3, the remaining one will be computed automatically

After setting up, the preview is shown

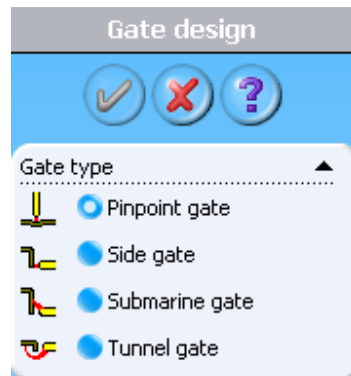


Click Create and the property tree will appear a extrude feature


4.Runner Pattern

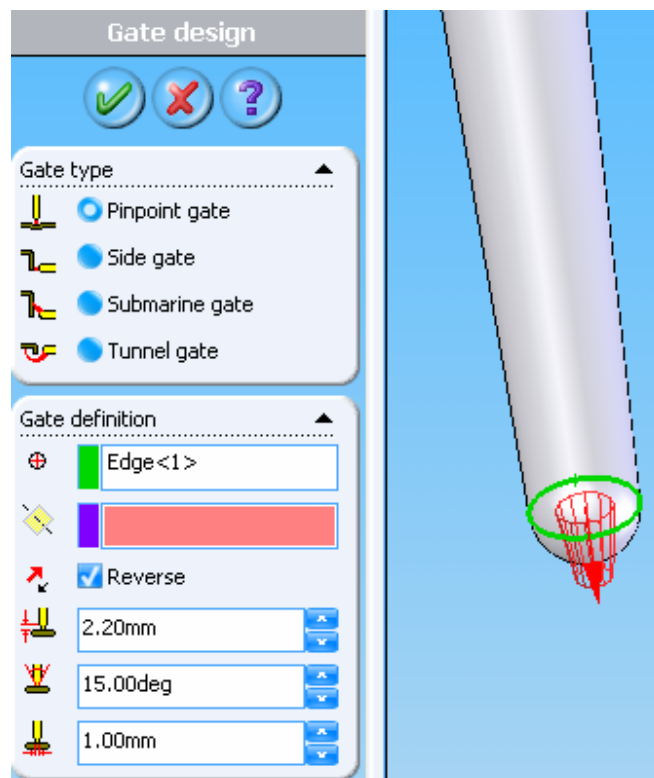
5.Gate design

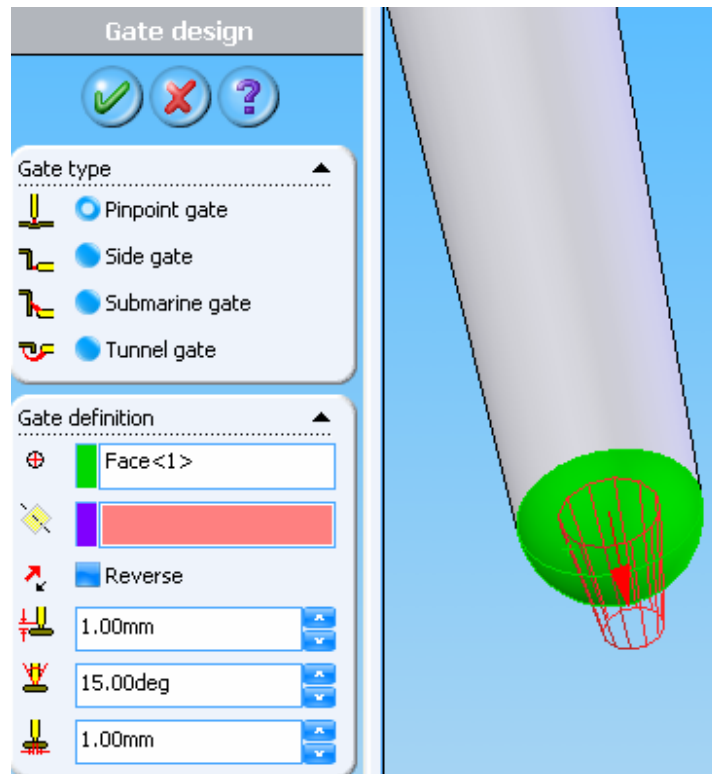
Four types of gate are available here : Pinpoint gate, Side gate, Submarine gate and Tunnel gate.








1. Pinpoint gate

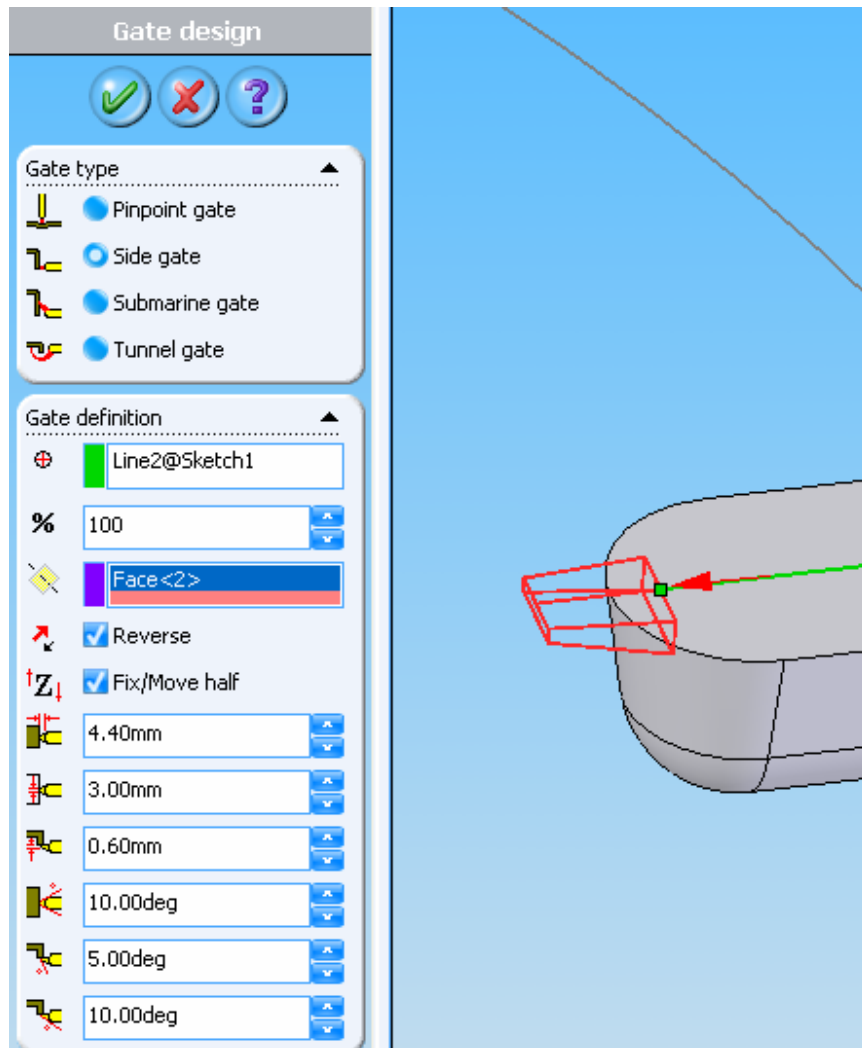
 : Select line or face to define the position of the gate.
















-  : select a plane or a line to define the direction
-  : Flip the direction of the gate
-  : define the length of the gate. If a circular edge or planar face is selected, this measure is the extension from the plane or circle center. If a spherical face is selected, the measure is the extension from the top of the curved surface.
-  : define the taper angle of the gate
-  : diameter of the gate

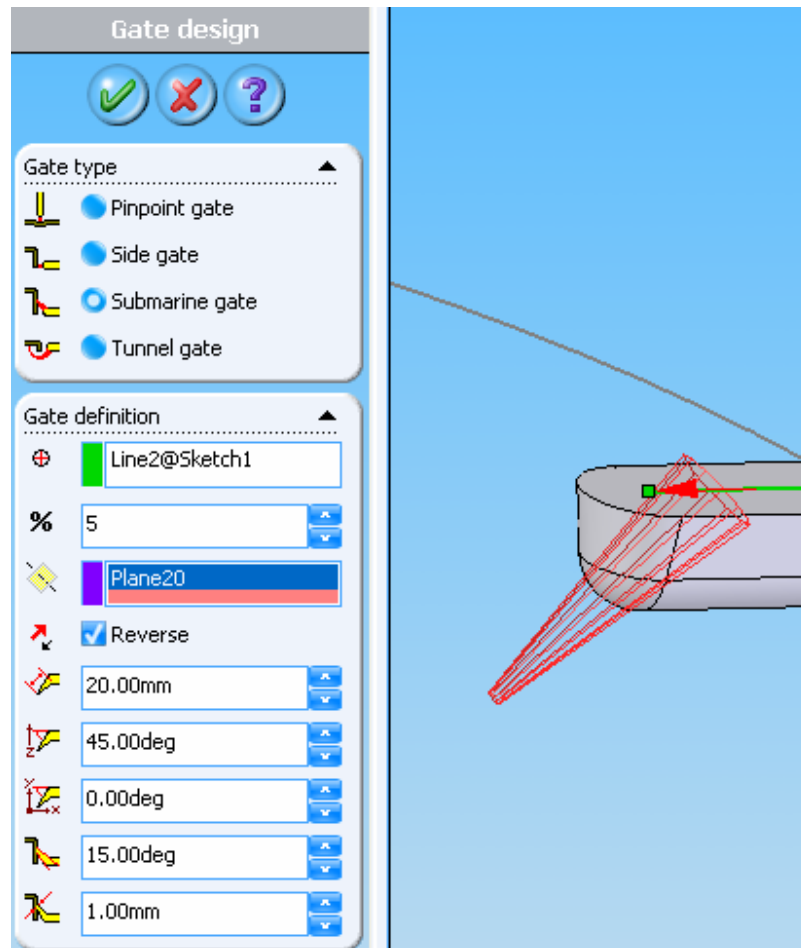
2. side gate





-  : select a sketch point or sketch segment
-  : adjust the the position ratio of a point on the sketch segment. This is activated when a sketch segment is selected
-  : select a reference plane or a linear edge to define the direction
-  : flip the direction of the gate
-  : Fix/Move half : change the gate location, it could be on fixed half or moving half
-  : define the lenght of gate
-  : define the width of gate
-  : define the thickness of gate
-  : approaching angle A
-  : approaching angle B


: approaching angle C

3. Submarine gate




: select a sketch point or sketch segment


: adjust the the position ratio of a point on the sketch segment. This is activated when a sketch segment is selected


: select a reference plane or a linear edge to define the direction


: flip the direction of the gate

: define the length of the gate

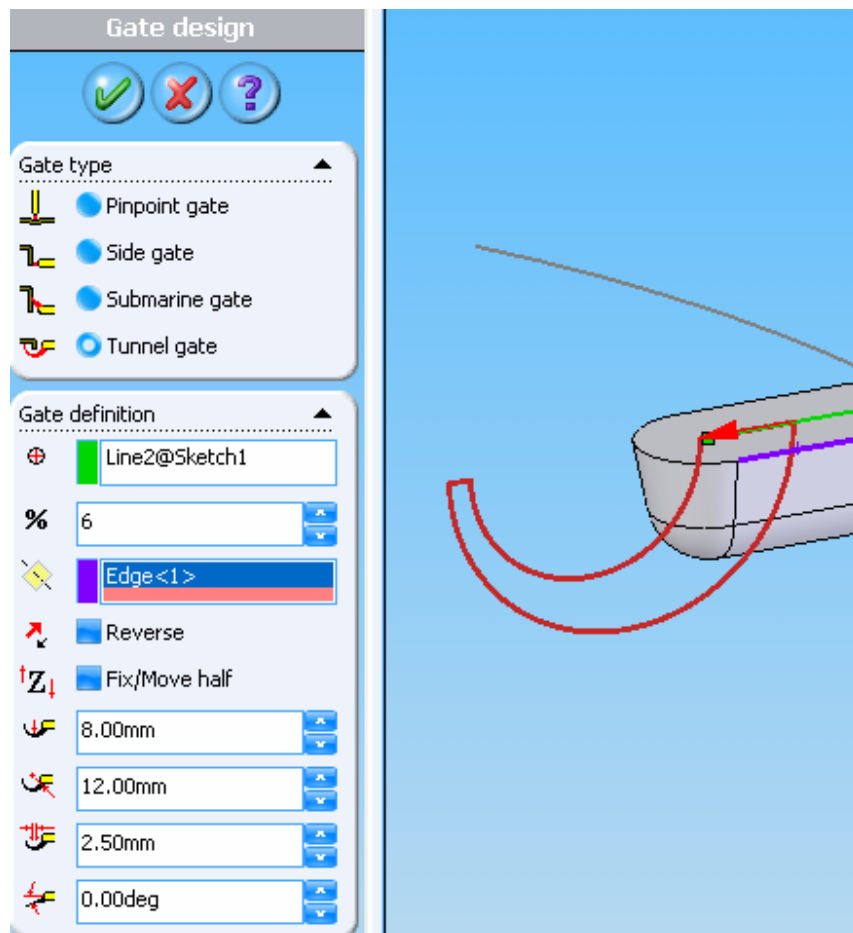
: define the angle between the gate and XY plane










: define the angle between gate and runner projected in Z direction

: approaching angle

: diameter of submarine gate

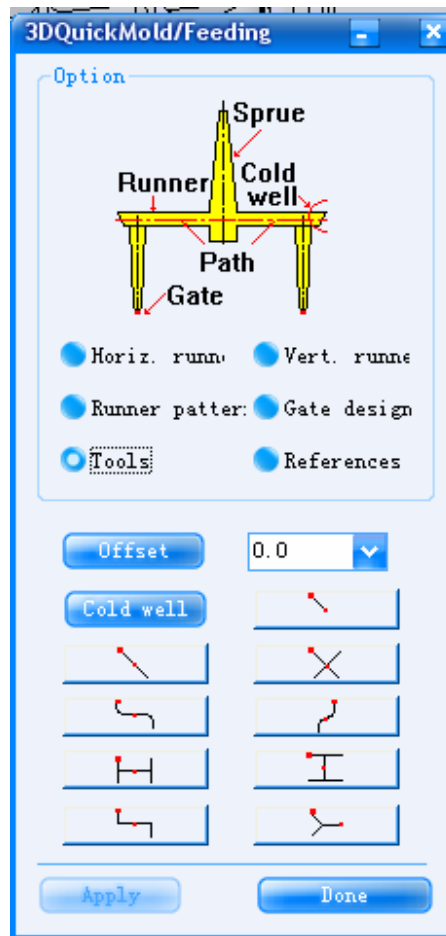
4. Tunnel gate



-  : select a sketch point or sketch segment
-  % : adjust the the position ratio of a point on the sketch segment. This is activated when a sketch segment is selected
-  : select a reference plane or a linear edge to define the direction
-  : flip the direction of the gate
-  **Z** : Fix/Move half : change the gate location, it could be on fixed half or moving half
-  : inner radius
-  : outer radius
-  : offset between two circular center
-  : define the angle between gate and runner projected in Z direction

6. Tools

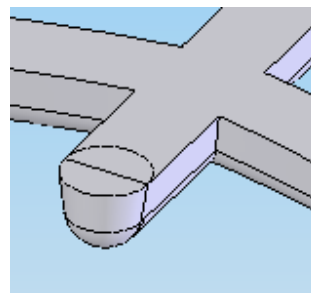
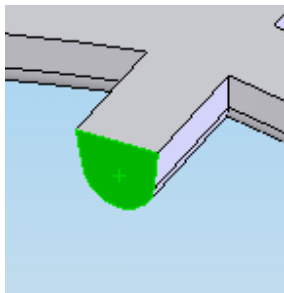
Some effective tools to define the runner path or modify runners

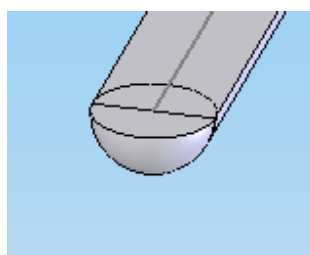
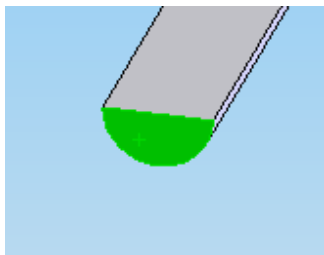
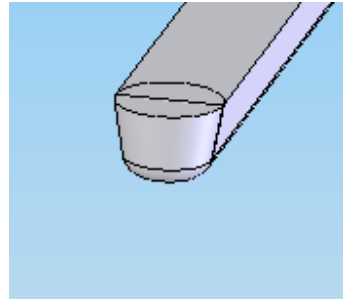
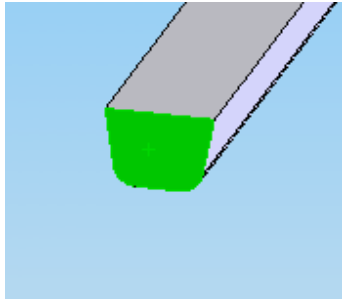
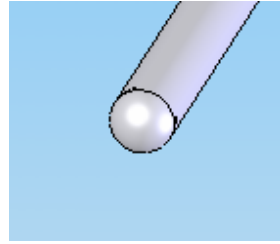
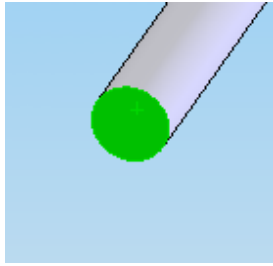


For all effective tools, pre-selections are needed to perform those functions, there is no PMP style interface will pop up.

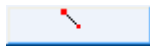
Offset : Select the face required to extend or shorten, set the offset distance, click this icon to perform.

Cold well : Select the end faces on runner that will add cold well on them, click this icon to perform. Cold wells of Runner with different cross-section are shown as follows.

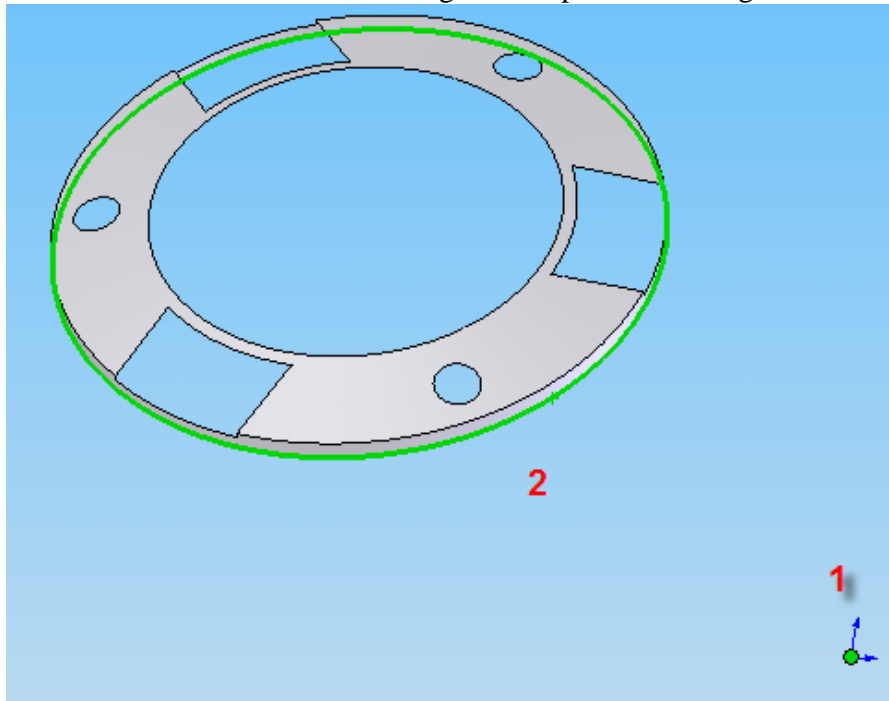




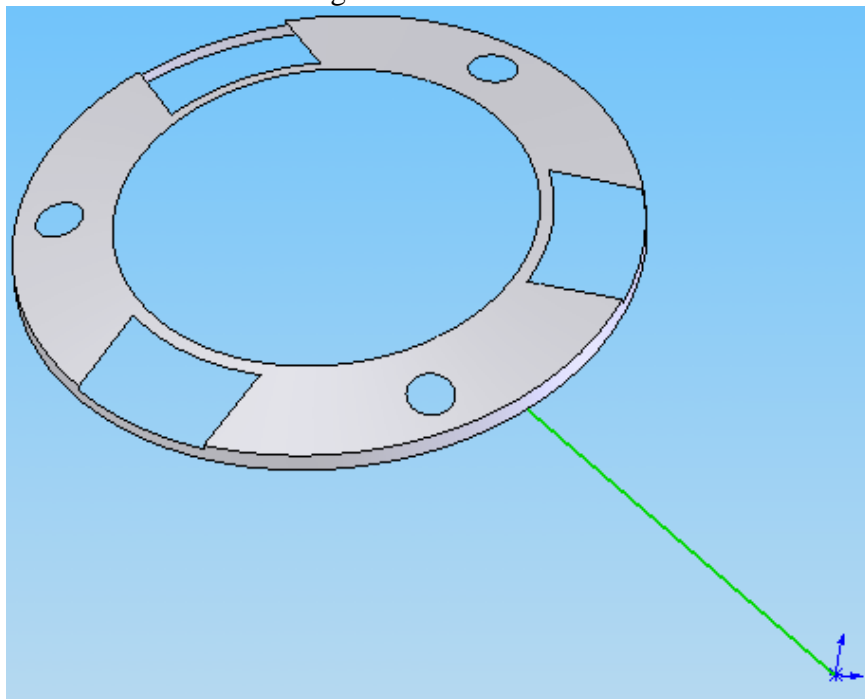
The functions below are for the quick build of runner path.



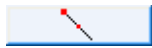
: Select as shown below the origin and a point on an edge.



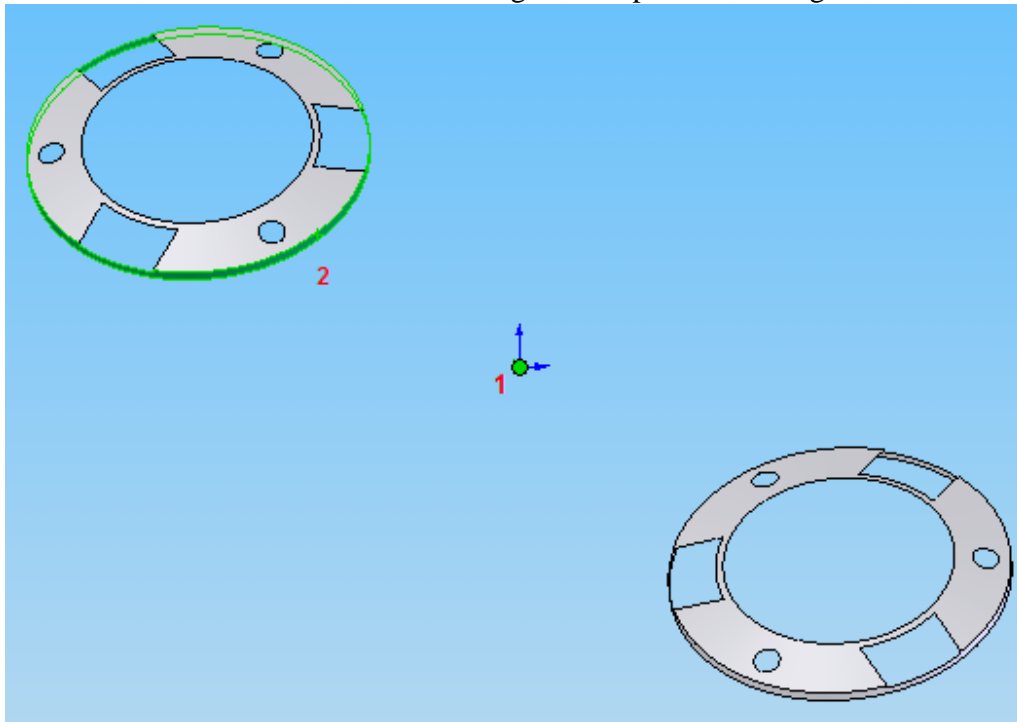
Click the icon to obtain the following sketch



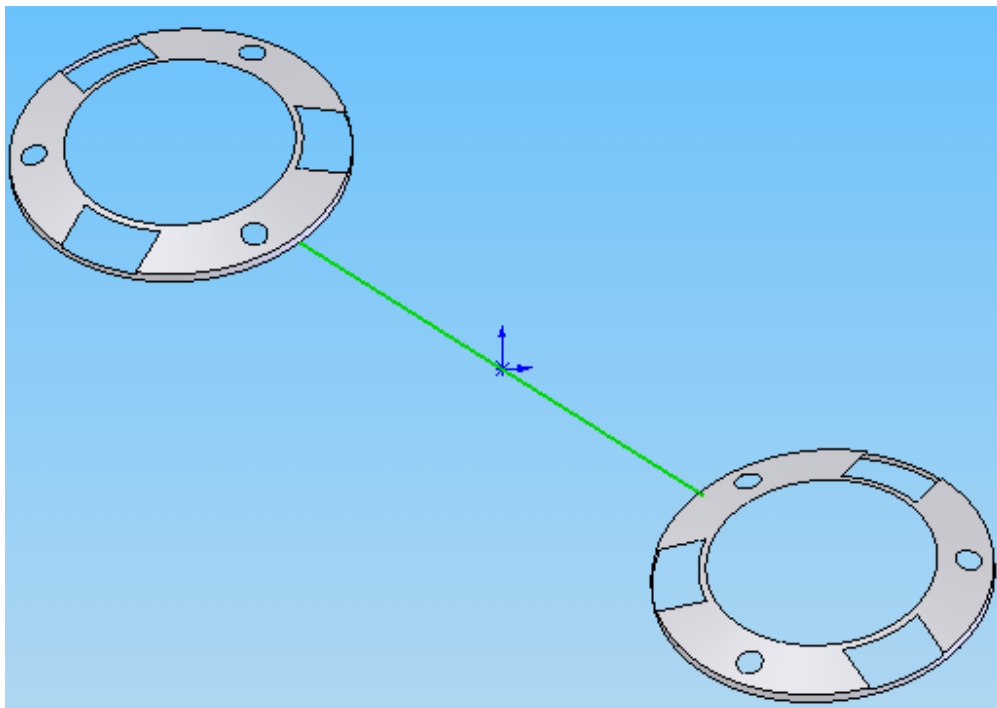
The line obtained is the projected line by the the two point line on XY plane



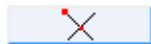
: Select as shown below the origin and a point on an edge.

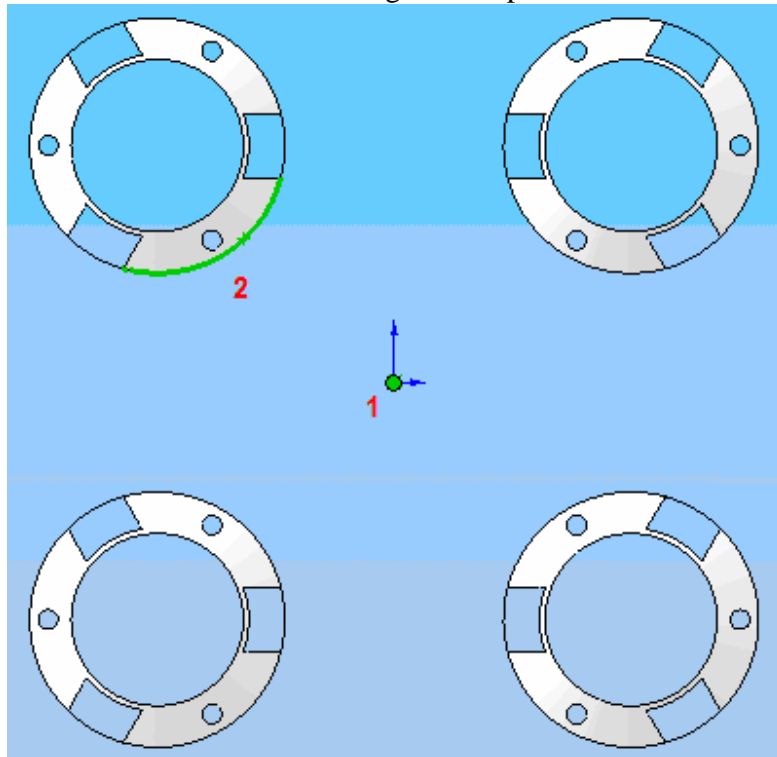


Click the icon to obtain the sketch.

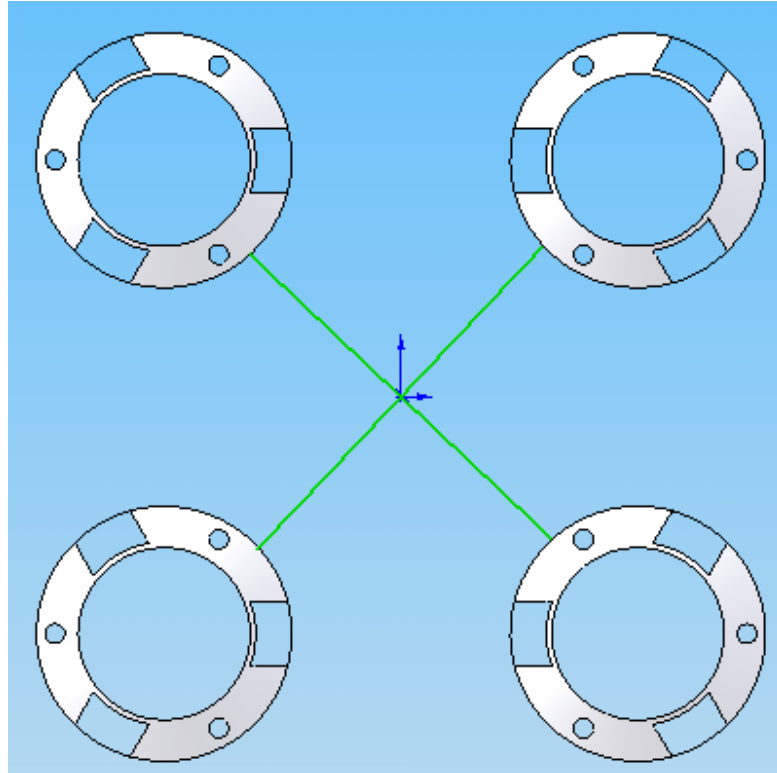


The Origin is the center point of the line obtained.

: Select as shown below the origin and a point on the curve.



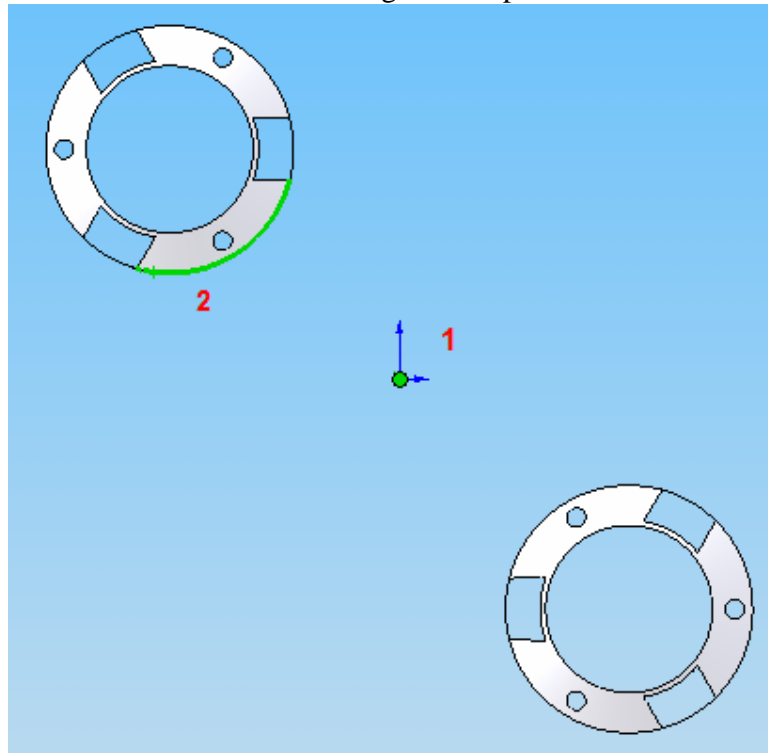
Click the icon to obtain the sketch.



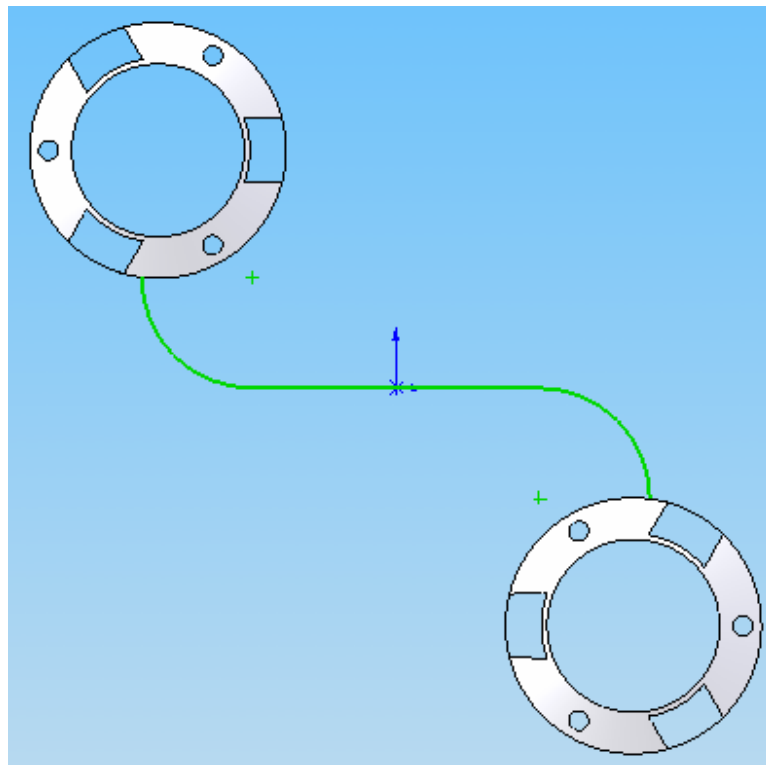
The two intersected lines are equal with the Origin as the center



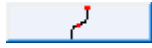
: Select as shown below the origin and a point on the curve.



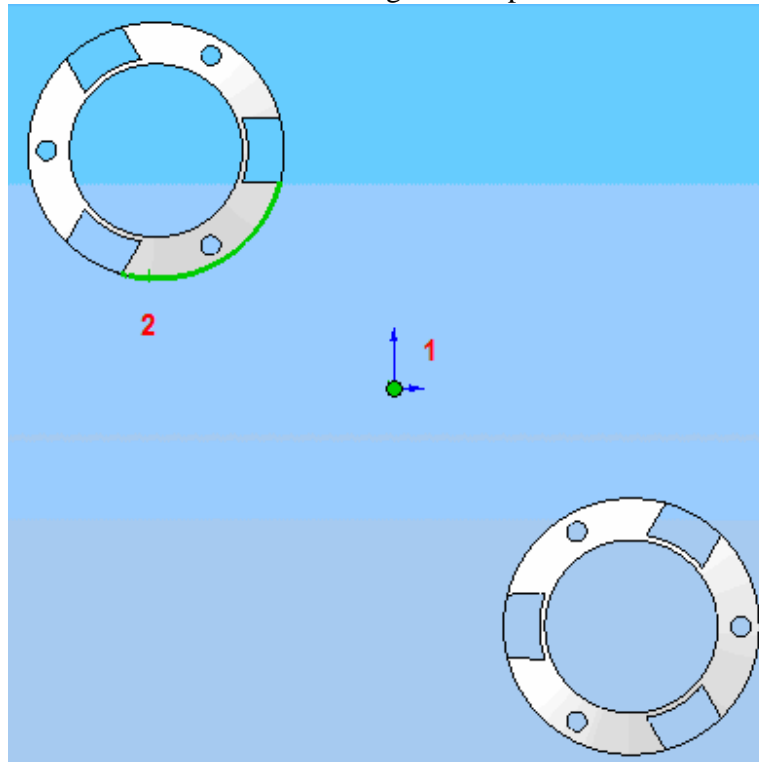
Click the icon to obtain the sketch.



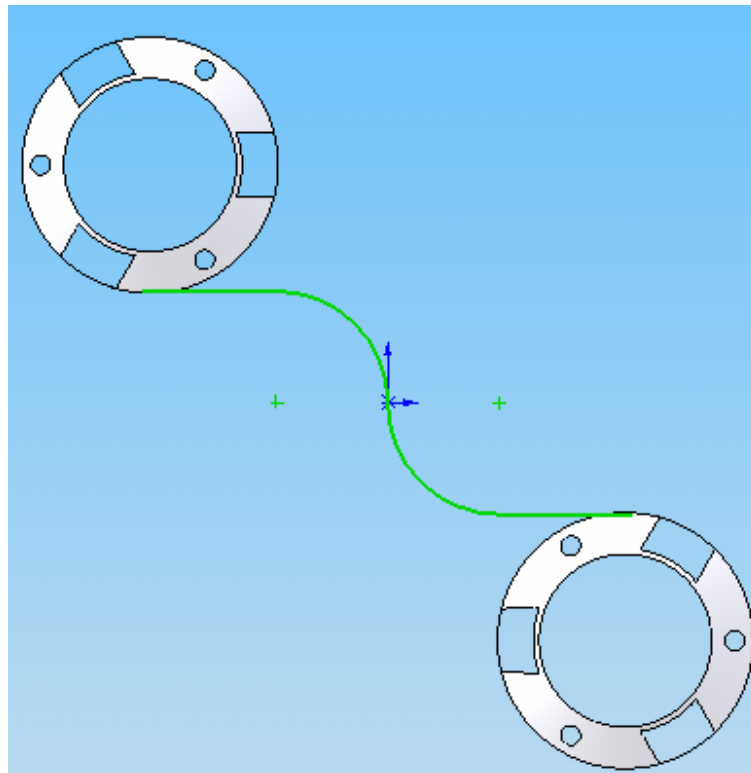
The sketch consists of a line and two arcs, and symmetrical about the Origin.



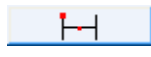
: Select as shown below the origin and a point on the curve.

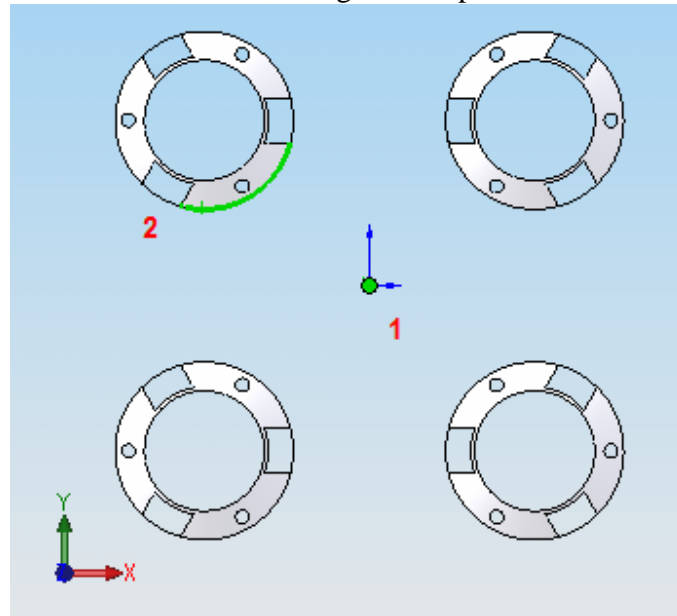


Click the icon to obtain the sketch.

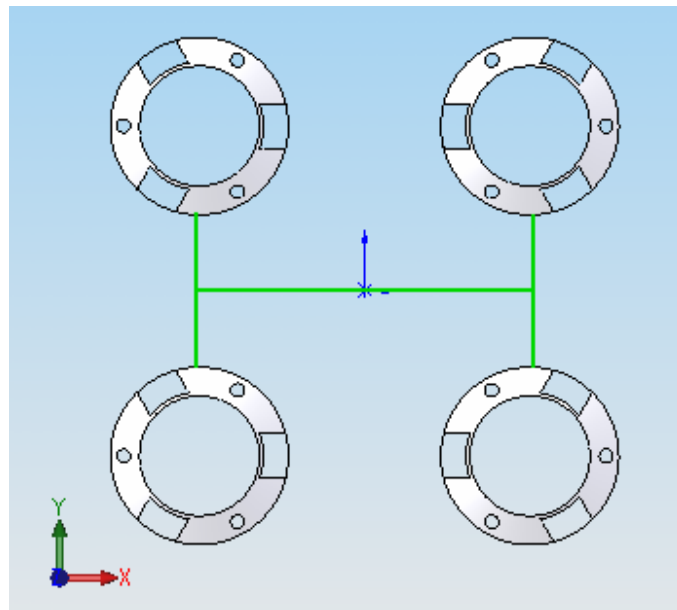


The sketch consists of two lines and two arcs, and symmetrical about the Origin.

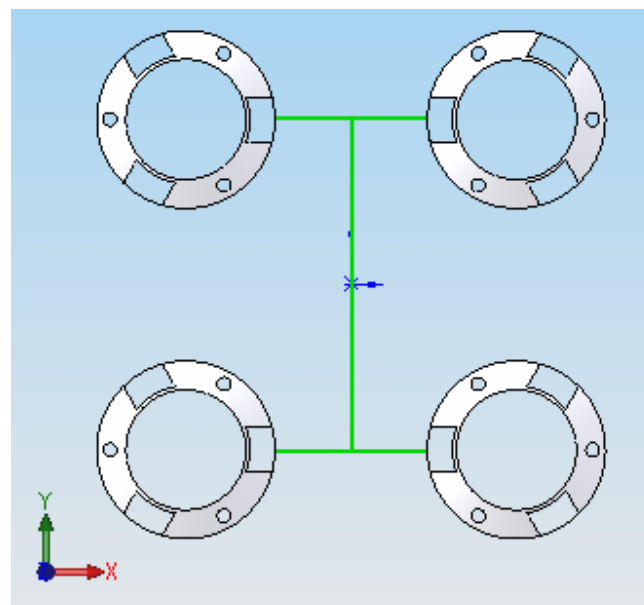
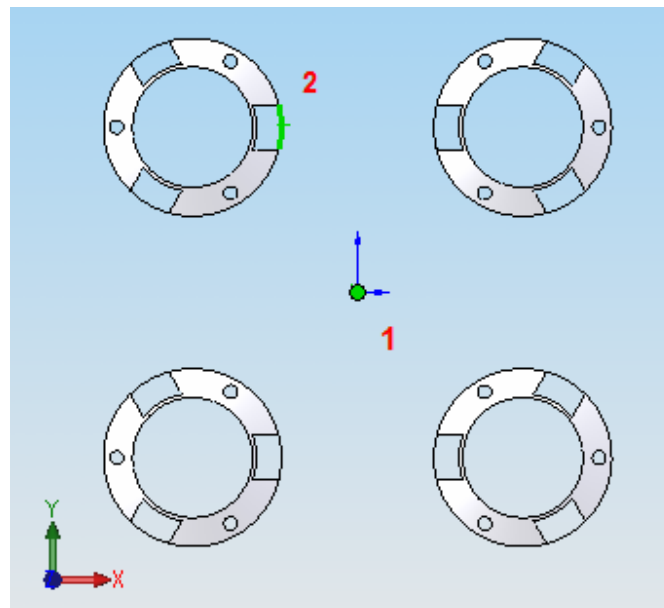
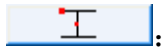
: Select as shown below the origin and a point on the curve



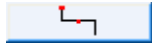
Click the icon to obtain the sketch.



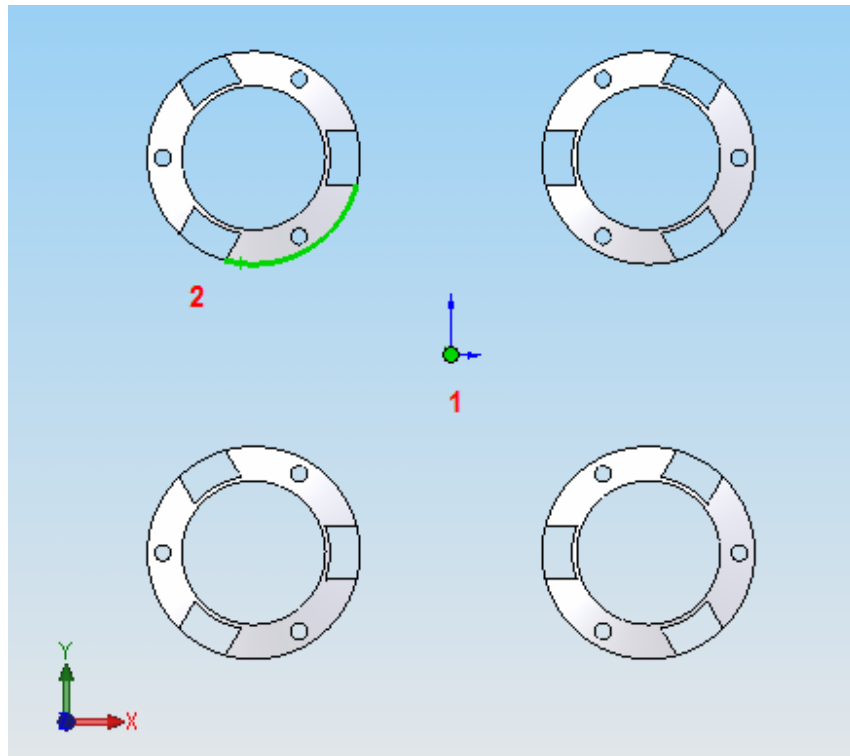
The sketch consists of 3 lines. When the view is changed to front view, the sketch displayed is same as the icon.



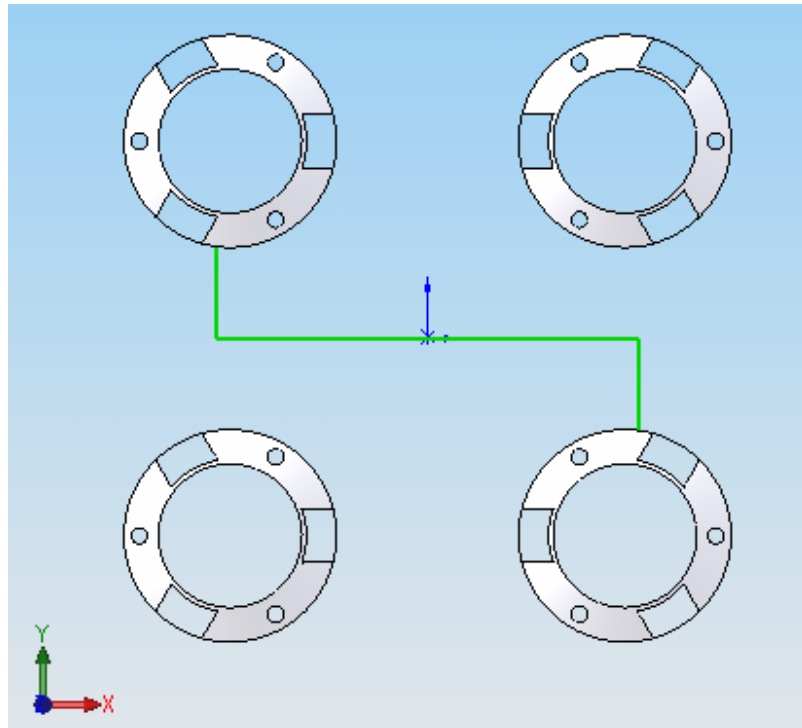
The sketch consists of 3 lines. When the view is changed to front view, the sketch displayed is same as the icon.

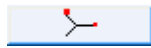


: Select as shown below the origin and a point on the curve.

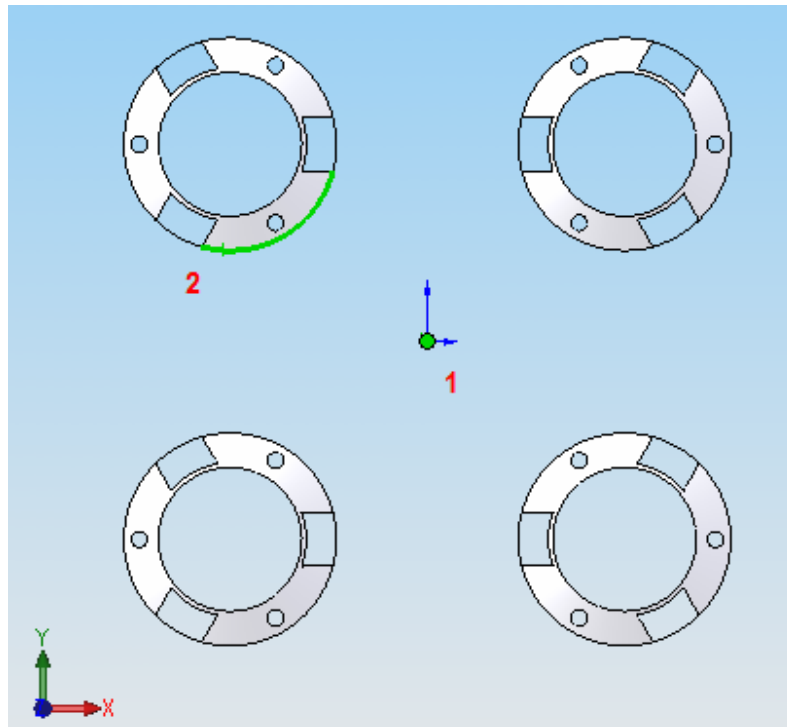


Click the icon to obtain the sketch.

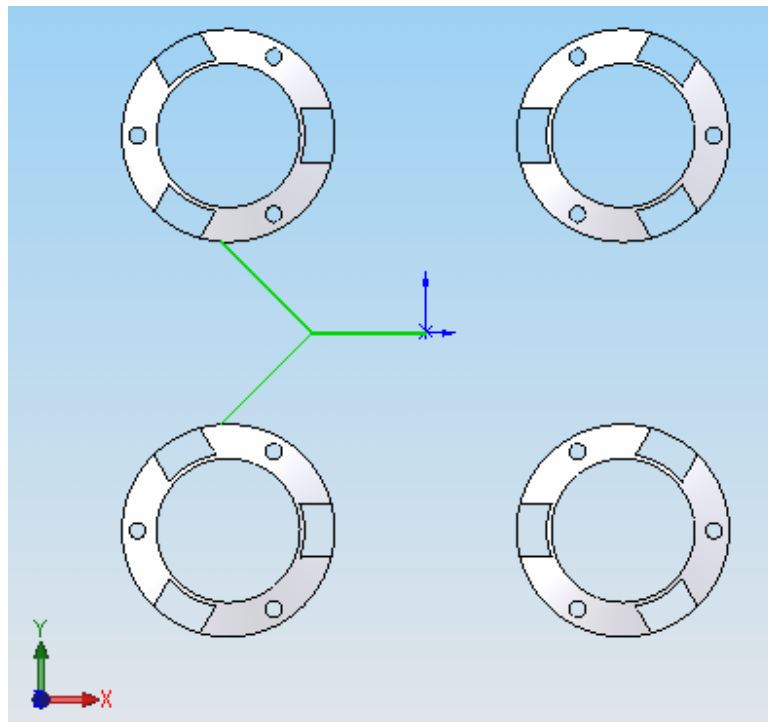




: Select as shown below the origin and a point on the curve.



Click the icon to obtain the sketch.



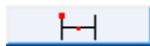
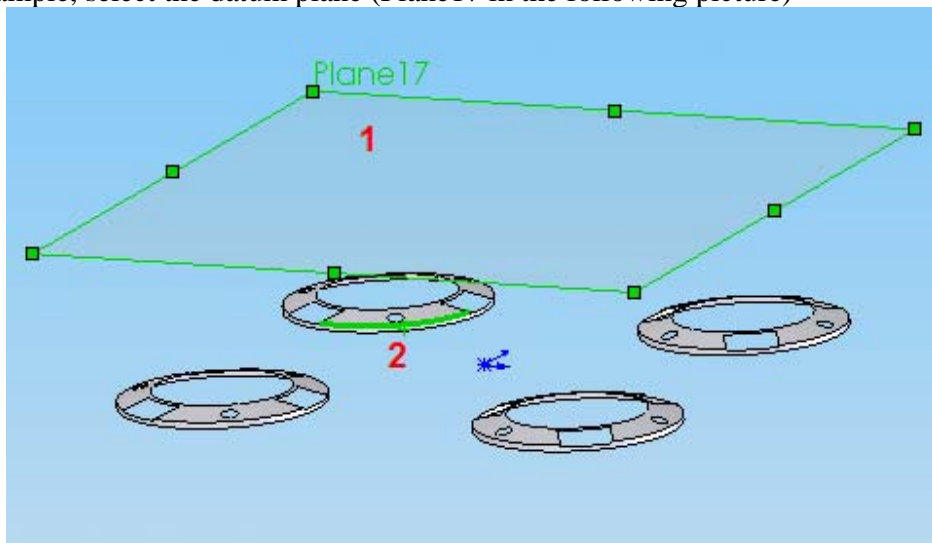
Please note that:

If the first selection is the Origin, a new sketch will be created.

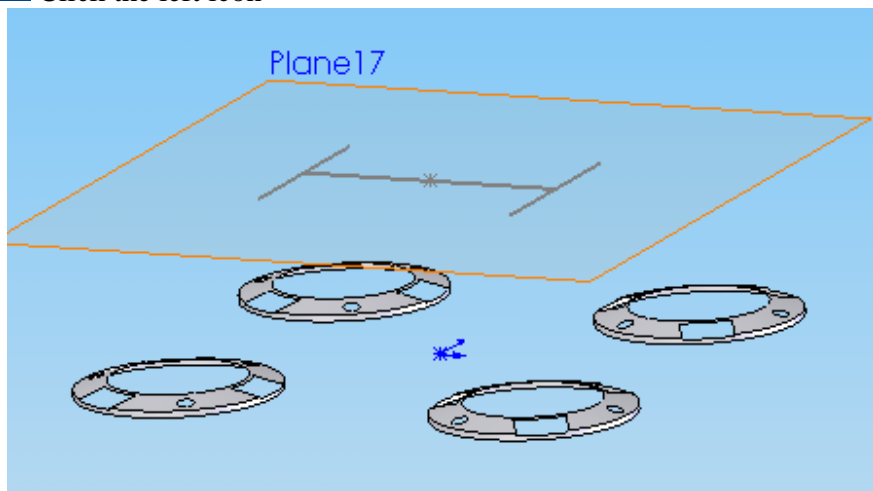
If the first selection is a sketch point, the new sketch segment will be created on the existing sketch where the sketch point lies on.

If such sketch is to be built on other reference plane, the first selection should be the reference instead of the Origin.

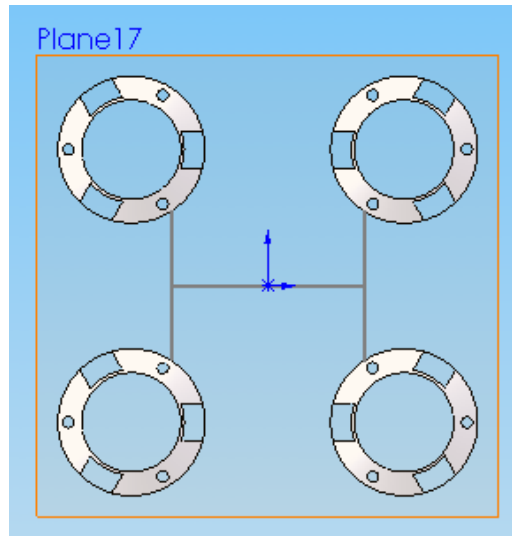
For example, select the datum plane (Plane17 in the following picture)



Click the left icon



The sketch obtained is built on the selected reference plane.

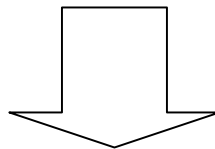


13.Electrode Manager

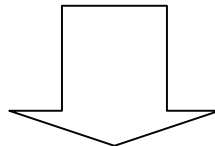
Electrode Manager is usually used after mold split. It is for the design of electrode.

The electrode design flow

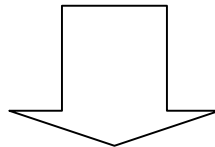
Derive part from core, cavity or side core (it could be any kind of part)



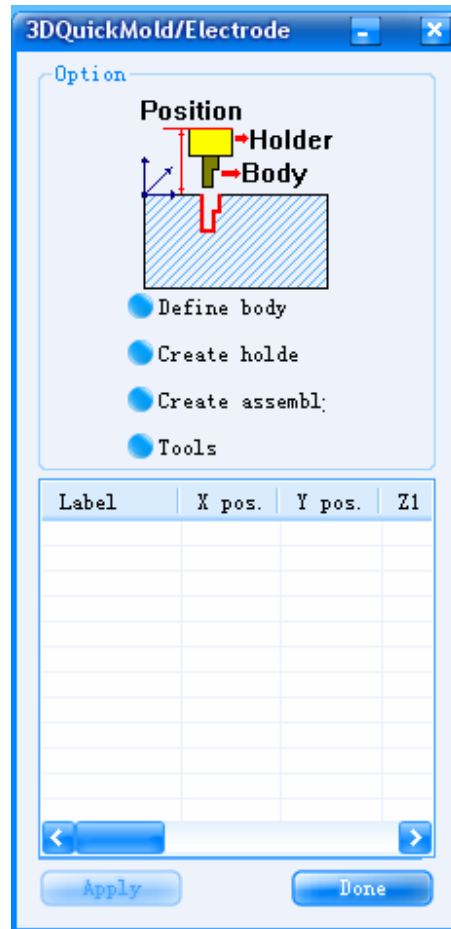
Create the electrode body



Create the holder



Create an electrode assembly



Select the desired option, click **Apply** to enter PMP style interface, click **Done** to terminate the electrode design.

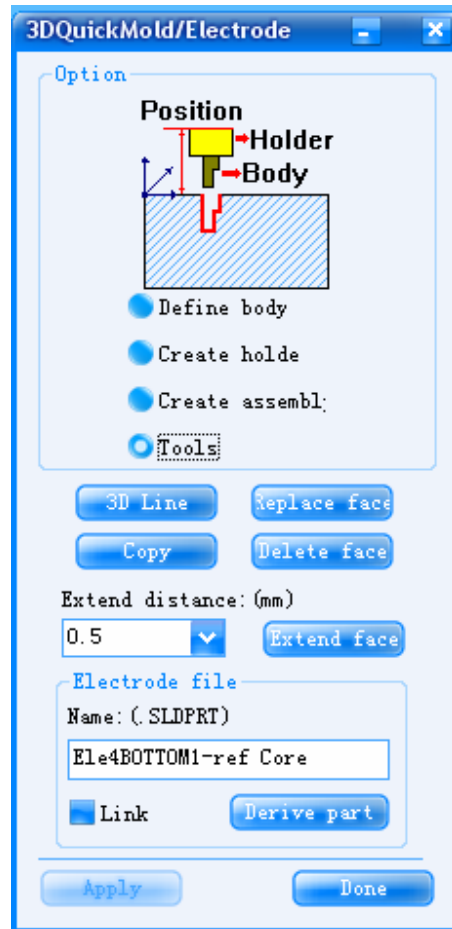
Reporting area at the bottom of the dialog will display the electrode's information such as position, height.

1. Tools

Some effective tools for electrode design at the bottom of the dialog, electrode file

Derive part: Derive Part

To avoid creating the electrodes on the core/cavity part directly, we need to create a new Solidworks part with the core or cavity inserted. This way, the associativity could be maintained and the total mold assembly won't become very complex.

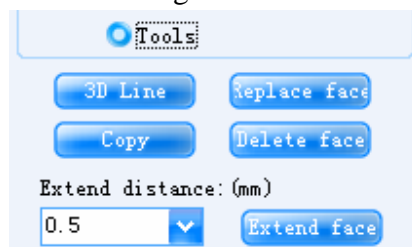


Name: Name for the the new part

Link: Specify if the external reference of the new created part is locked or not.

- Checked REF Cavity ->
The feature in the tree indicate that the external reference is unlocked.
- Unchecked REF Cavity -> *
The feature in the tree indicate that the external reference is locked.

The icons on the upper part of the dialog are some tools for electrode design



: Select 2 vertice to build a 3D line, it is a Solidworks 3D curve.

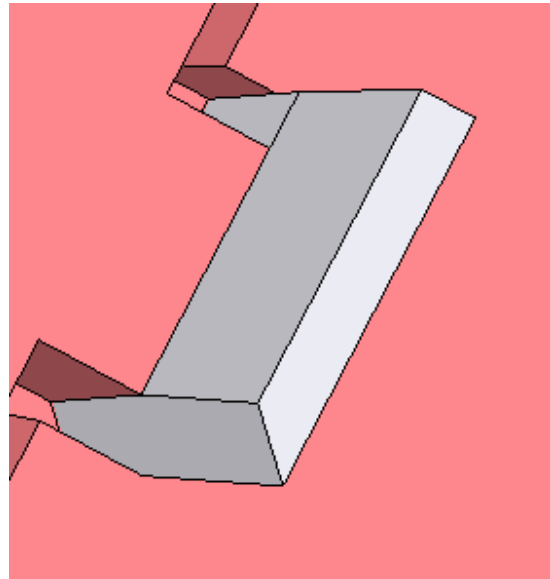
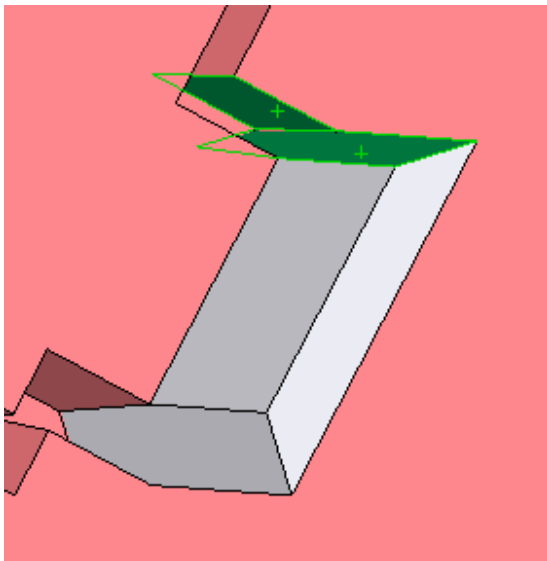


Two vertices

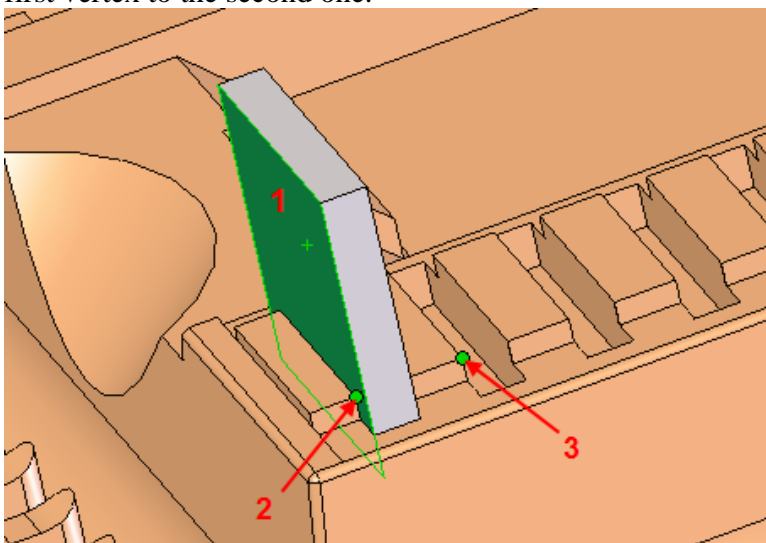


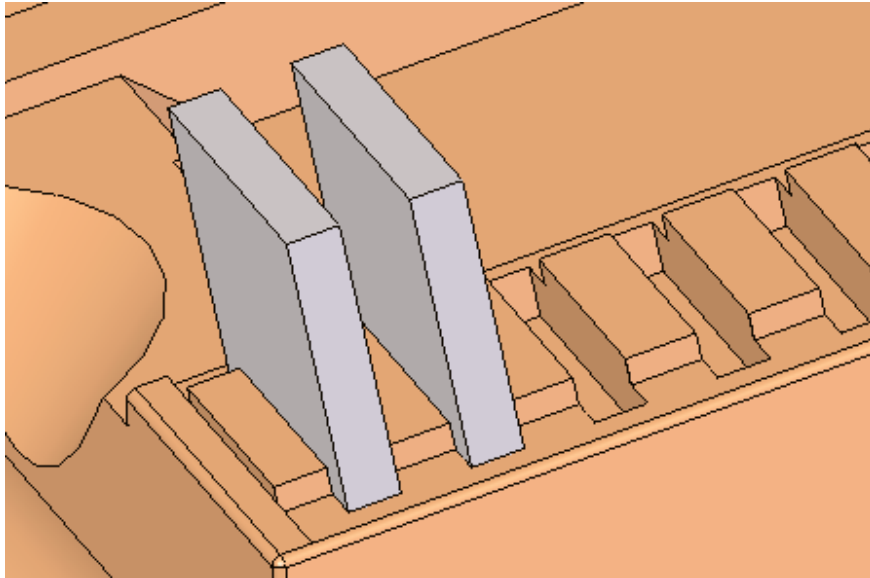
a 3D curve

Replace face: Select the face on the electrode to be changed and a face on the core or cavity, click this button the face on the electrode is replaced by the selected face on the core or cavity.



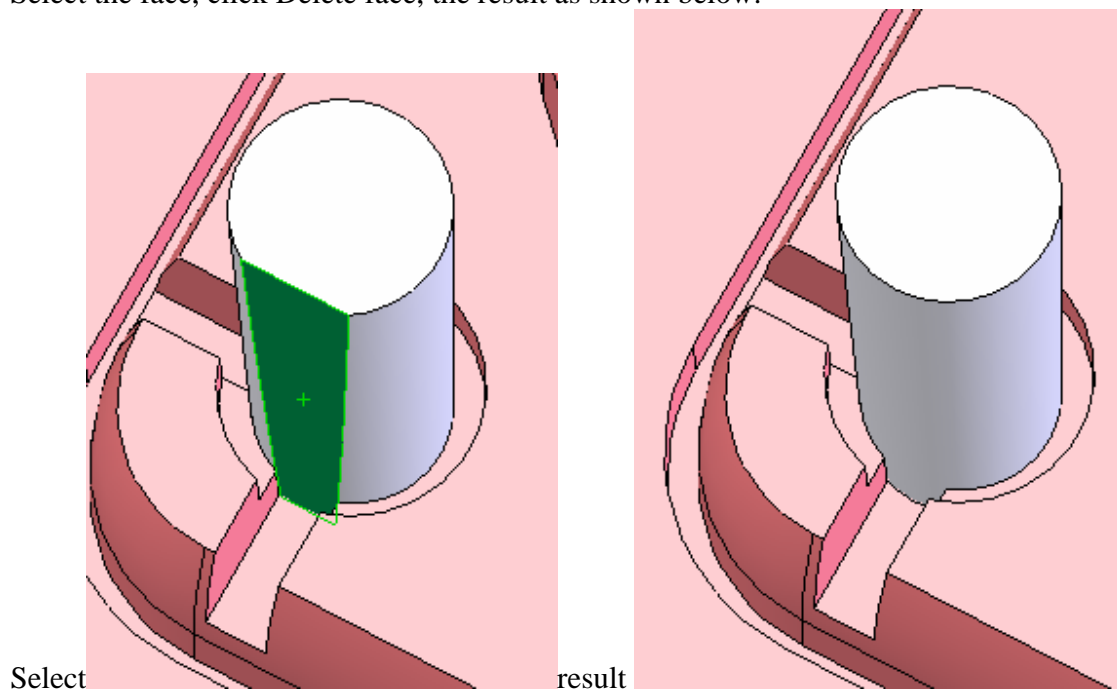
Copy: Copy body. Select a face on the existing electrode body and two vertices on the electrode part in order, click this button, electrode body will be copied from the first vertex to the second one.

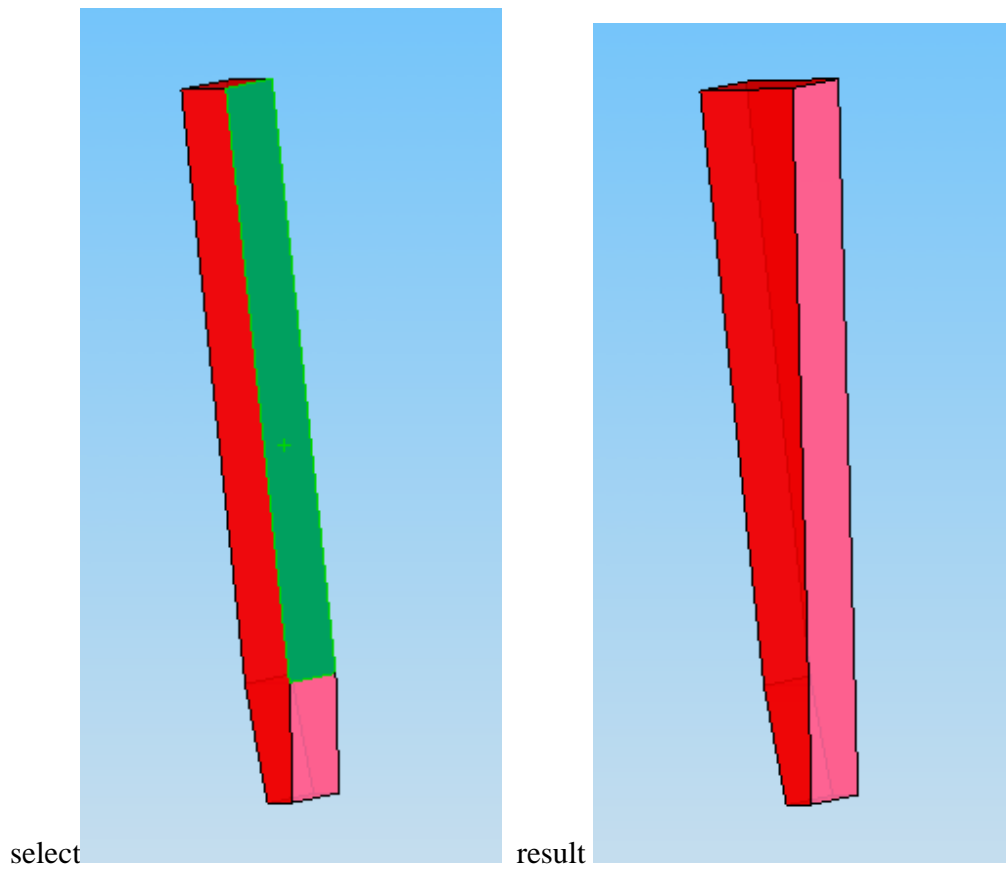




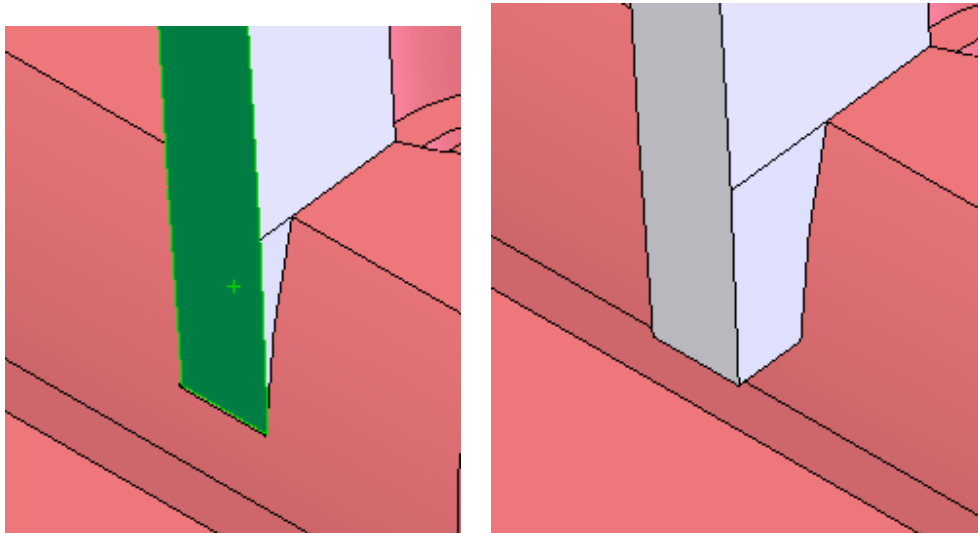
Delete face: A similar function like the Solidworks one. It is put here as this function is often used in electrode design.

Select the face, click Delete face, the result as shown below.

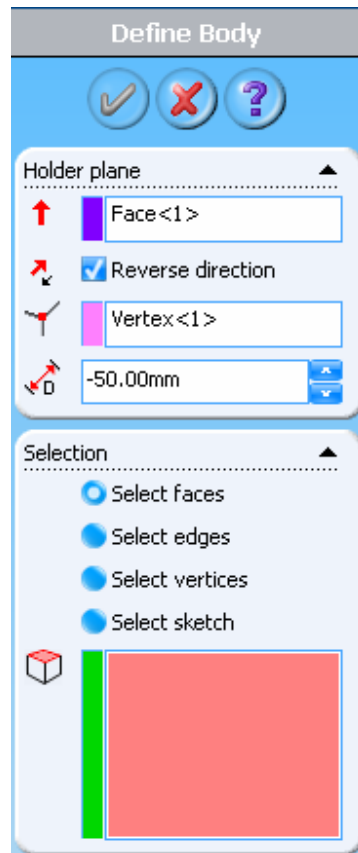




Extend face: Extend the selected face. Offset value could be set.



2. Define body





Holder plane: Create sketch plane to define the electrode body


There are two methods to obtain the sketch plane:

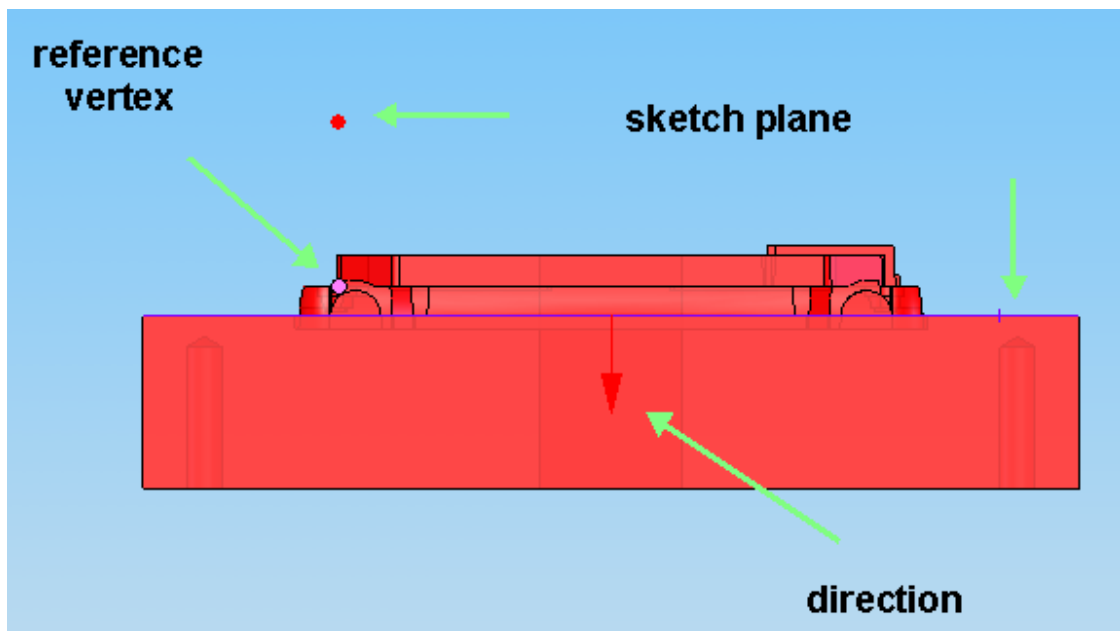
- 1, Pass through a vertex and parallel to a reference plane or planar face.
- 2, Parallel to a face or reference plane at a defined distance.

 Direction: Select a planar face or reference plane to define the EDM direction.

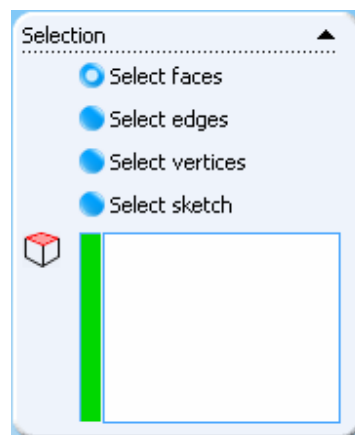
 Reverse direction: Reverse the direction, if the EDM direction is not the default one.

 Select reference vertex: Select reference vertex to define the offset value of the holder plane

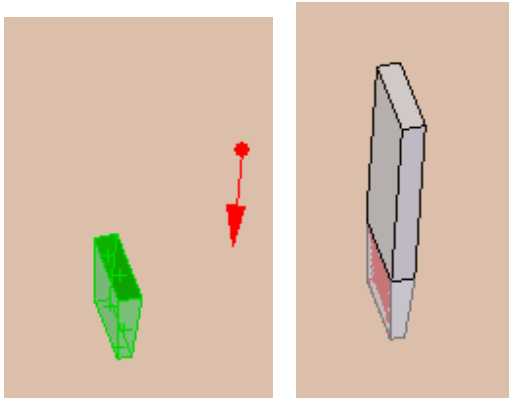
 Distance: Input a distance to define the sketch plane, there is a red dot on the screen to represent the holder position, this position must ensure that the electrode's holder won't be interfere with the part surface.



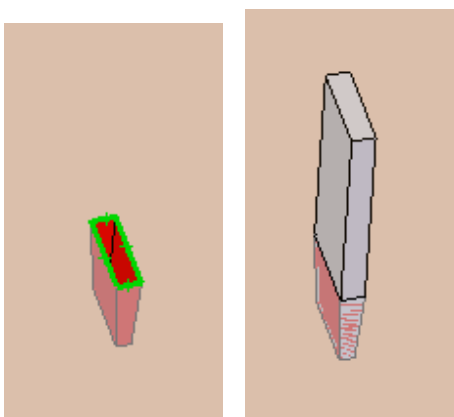
Selection:



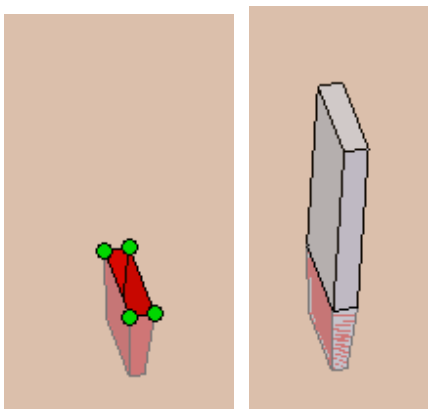
Select faces: Select some faces on the part to be the reference to create electrode.



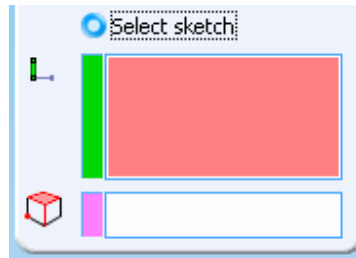
Select edges: Select some edges on the part to be the reference to define the EDM area. Those edges should be formed a closed area.





Select vertices: Select some vertices on the part to define the electrode's profile.

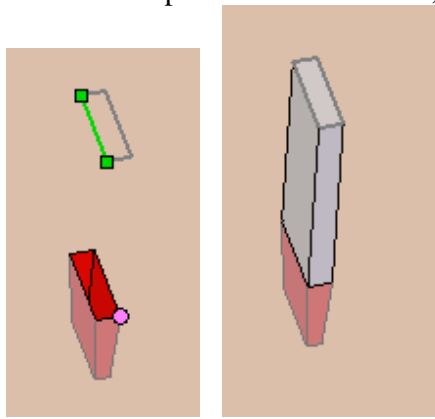


Select sketch: Select sketch on the part to define electrode's profile directly.



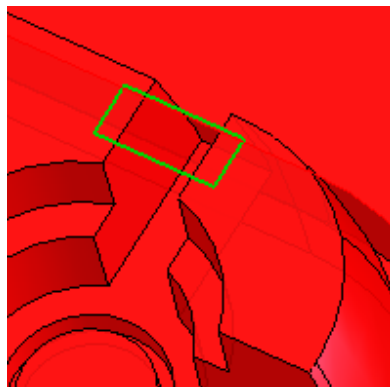
 : Select the sketch segment, they should be a closed loop

 middle point : Select vertex, edge or face here to define the electrode body

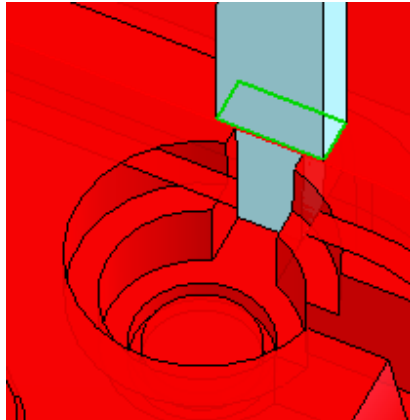


Example to explain the difference in selecting vertex, edge or face

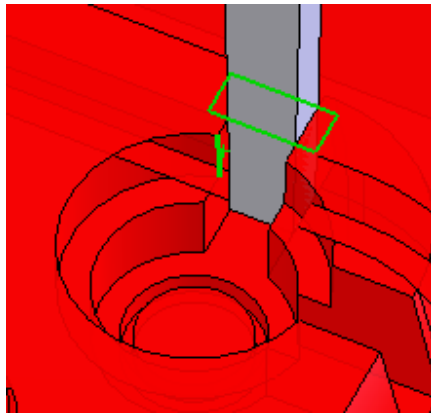
A sketch used to define the electrode body



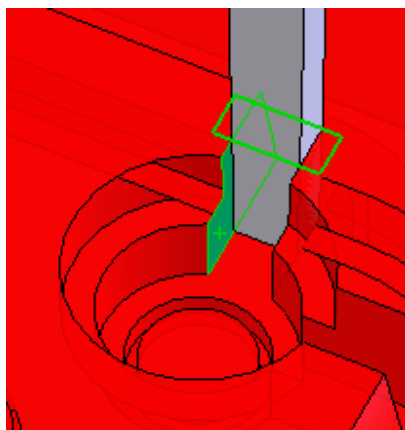
Select a vertex on the part, Extrude feature will extend from the sketch plane to the body where the vertex belongs to.



Select edge:



Select face:



3.Create holder

Create the holder of the electrode, the holder could be circular or rectangular type.

Rectangle: Create a holder with rectangular shape.

Select base face: Select the base face to be the reference plane to create the holder. Its minimum size is determined and displayed automatically

Reverse direction: Reverse the direction of the holder if necessary.

Favorite size: Select the standard or customized holder size available in the system


Block Electrode.xls could be found under the installation folder, this file could be customized.

	A	B	C	D
1		H	V	L
2	Customer	10	10	10
3	EHB-5-5	5	5	10
4	EHB-5-10	10	5	10
5	EHB-5-15	15	5	10
6	EHB-5-20	20	5	10
7	EHB-5-25	25	5	10
8	EHB-5-30	30	5	10

L: Define the length of holder

W: Define the width of holder


H: Define the height of holder


: Define the angle of rotation of the holder


Electrode Label: Electrode label name

Electrode Name: 3DQuickMold will add default name to the electrode if it is not specified.

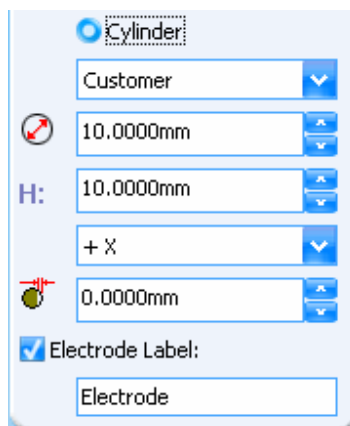
Reference vertices: The default holder center is the center of the selected face. However, it can be defined by specifying a reference vertex and its relative position.


: Select a coordinate system for the position of the electrode.

: the X coordinate of the centre of the electrode holder relative to the above selected coordinate system

: the Y coordinate of the centre of the electrode holder relative to the above selected coordinate system

Cylinder: Create a cylindrical holder

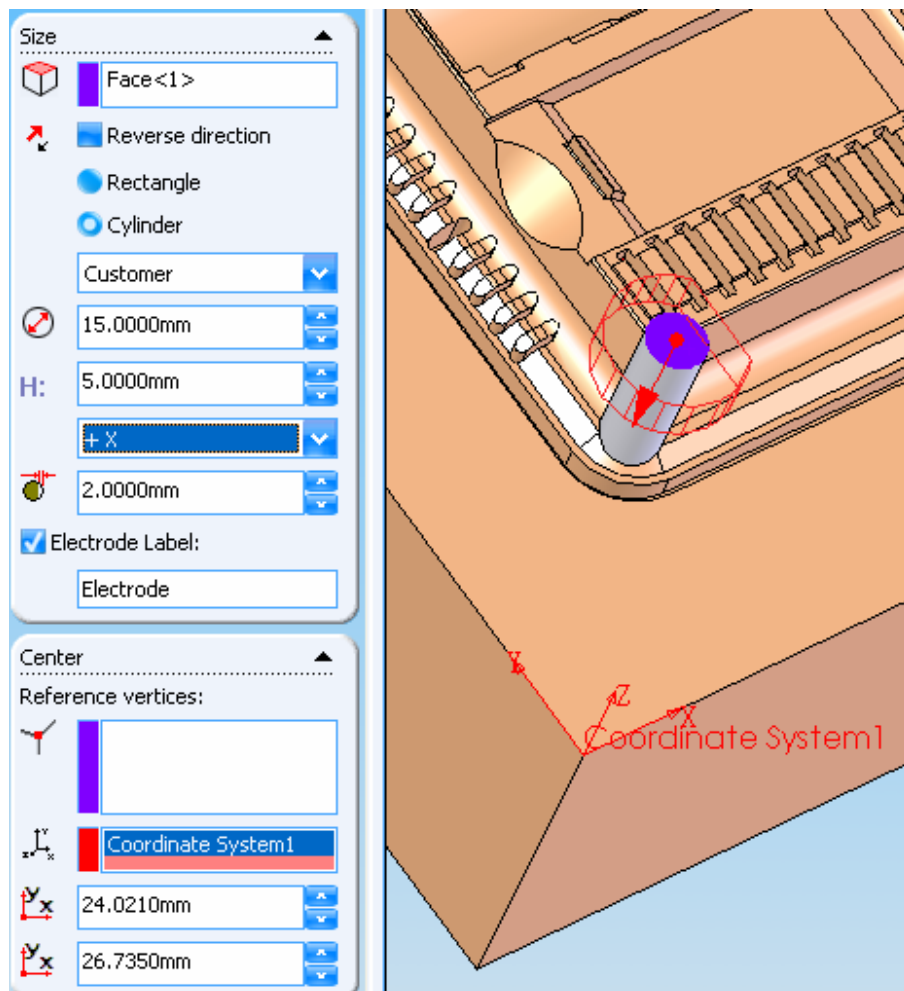


: Define the diameter of the holder.

H: : Define the height of the holder

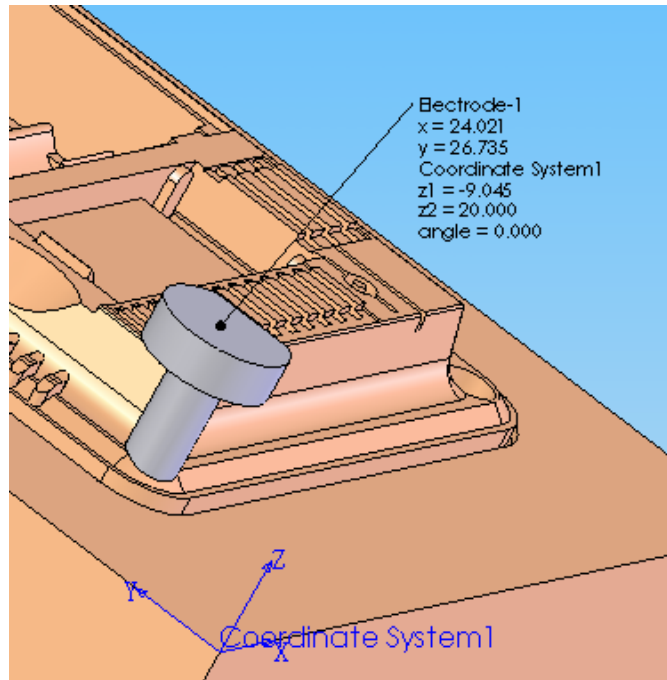
Mark Side: The orientation of the mark area

: Offset distance

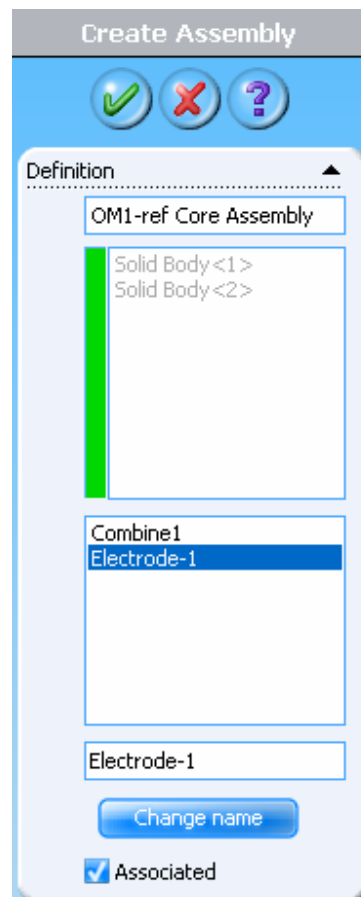


Click OK, an electrode holder is created. There is an annotation attached to the holder, major electrode's information is displayed on the screen such as its name, coordinate system and positions.

Z1 and Z2 are the electrode's lowest and highest position value in the selected coordinate system.



4.Create Assembly



Save the electrode part with multiple bodies into an assembly.

Assembly name: Input the name of the assembly to be created

Body list: List out all shown solid bodies in the electrode part

Select electrode: Select a solid from the shown body list

Part name: Input the component name. By default, the component's name is the solid body's name in the electrode part.

Change name: Change the selected body name to the input one below.

Associated: If this option is unchecked, the imported body will be created in the assembly component. In a certain situation, the imported bodies could achieve a better performance.

3 D QuickTools Limited



3D QuickTools Limited - All right reserved 2008

Unit 115, No.5 Science Park West Avenue,
Hong Kong Science Park, Shatin, N.T, Hong Kong
Tel: +852 2788 2832 | Fax: +852 2777 6676 | www.3dquicktools.com